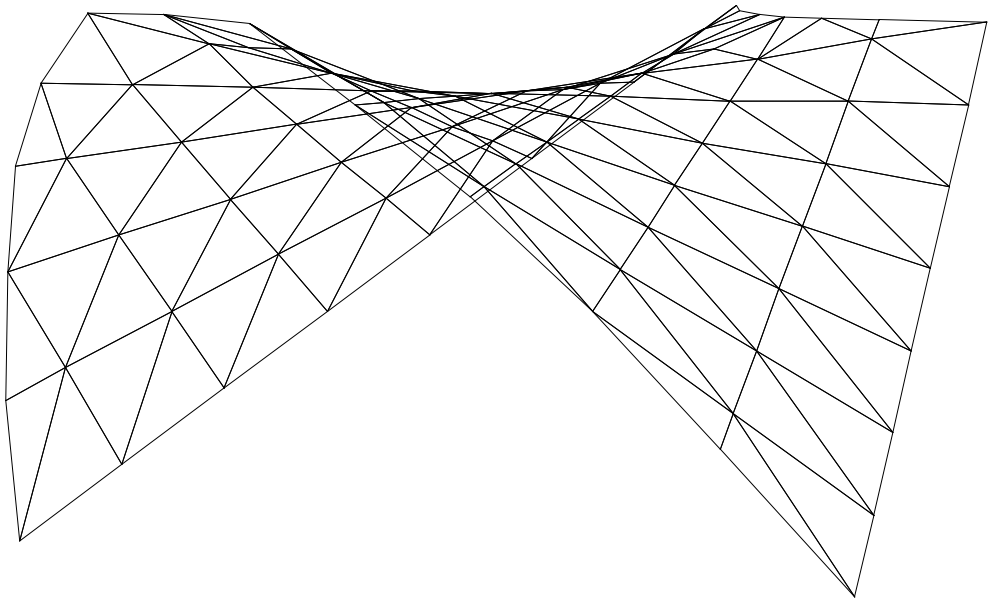


S T R A P

for

Windows

STRUCTURAL ANALYSIS
PROGRAM

USER'S MANUAL

Version 12.5

December 2006

** Blank page **

Disclaimer

The *STRAP* programs have been written by a team of highly qualified engineers and programmers and have been extensively tested. Nevertheless, the authors of the software do not assume responsibility for the validity of the results obtained from the programs or for the accuracy of this documentation.

The user must verify his own results

The authors remind the user that the programs are to be used as a tool for structural analysis, and that the engineering judgement of the user is the final arbiter in the development of a suitable model and the interpretation of the results.

Windows is a registered trademark of Microsoft Corp.

AutoCAD is a registered trademark of Autodesk Inc.

** Blank page **

Table of Contents

1 General 1.0-1

1.1 Installing the Program 1.1-1

1.1.1 Tutorial 1.1-2

1.2 Running the program 1.2-1

1.2.1 Define a new model 1.2-2

1.2.2 Amend/rerun an existing model 1.2-3

1.3 Main menu - "File" options 1.3-1

1.3.1 Print 1.3-2

1.3.2 File management 1.3-2

1.3.2.1 Delete 1.3-2

1.3.2.2 Make a copy of a model 1.3-3

1.3.2.3 Copy to another directory 1.3-4

1.3.2.4 Copy from another directory 1.3-5

1.3.3 Change current directory 1.3-5

1.3.4 Copy to a ZIP file 1.3-6

1.3.5 UnZIP a model 1.3-6

1.3.6 Display model file names 1.3-7

1.3.7 Add a file to model list 1.3-8

1.3.8 Recover model geometry 1.3-8

1.4 Setup 1.4-1

1.4.1 Setup – Colors 1.4-1

1.4.2 Miscellaneous 1.4-1

1.4.3 Print parameters 1.4-2

1.4.4 Toolbars 1.4-3

1.5 Utilities 1.5-1

1.5.1 Footing design 1.5-1

1.5.2 User steel section table 1.5-1

1.5.3 Compute section properties 1.5-1

1.5.4 Combine results of two projects 1.5-2

1.5.5 Recreate the model list 1.5-2

1.5.6 STAAD File Conversion 1.5-2

1.5.7 Create a STAAD geometry input file 1.5-2

1.5.8 Create steel detailing files 1.5-2

1.6 DXF options 1.6-1

1.6.1 Convert metafile to DXF 1.6-1

1.6.2 Convert STRAP file to DXF (DXF export) 1.6-1

1.6.3 Create a new model from DXF file (DXF Import) 1.6-1

1.7 Getting Started 1.7-1

1.7.1 Selection of the Computer Model 1.7-1

1.7.2 Inputting the Model 1.7-4

1.8 Coordinate systems 1.8-1
1.8.1 Global Coordinate System 1.8-2
1.8.2 Local Coordinate System 1.8-2
1.8.2.1 Beam Elements 1.8-2
1.8.2.2 Quadrilateral Finite Elements 1.8-4
1.8.2.3 Triangular Finite Elements 1.8-5
1.8.2.4 Wall elements 1.8-6
1.8.3 Sign Conventions - Forces and Moments 1.8-6
1.9 General options 1.9-1
1.9.1 Step 1.9-1
1.9.2 Dialog boxes 1.9-1
1.9.3 Shortcut Menus 1.9-3
1.9.4 Side Menus 1.9-4
1.9.5 Help 1.9-4
1.9.6 Icon bar 1.9-5
1.9.7 Undo 1.9-5
1.9.8 Print tables 1.9-6
1.9.9 Print drawing 1.9-6
1.9.9.1 Regular drawing: 1.9-6
1.9.9.2 Rendered drawing: 1.9-8
1.9.10 Print/edit drawing 1.9-8
1.9.11 Entities Selection 1.9-10
1.9.11.1 Node Selection 1.9-10
1.9.11.2 Beam/Element Selection 1.9-13
1.10 Command mode 1.10-1
1.10.1 Enter a Command 1.10-1
1.10.2 Revise a Command 1.10-1
1.10.3 Retrieve a command (from the Clipboard) 1.10-1
1.10.4 Command Format - General 1.10-3
1.11 Batch mode 1.11-1
1.11.1 Batch mode - Geometry 1.11-1
1.11.2 Batch mode - Loads 1.11-3
1.11.3 Batch mode - Combinations 1.11-4

2 Geometry - Main Menu 2-1
2.1 Nodes 2-2
2.1.1 Single Node 2-2
2.1.2 Line of Nodes - Equal Spacing 2-6
2.1.3 Line of Nodes - Unequal Spacing 2-7
2.1.4 Nodes - Coordinate system 2-8
2.2 Restraints 2-11
2.3 Beams 2-12
2.4 Elements 2-13
2.5 Springs 2-15
2.6 Copy 2-16
2.7 Solid elements 2-17

2.8 Wall Elements 2-18
2.9 Sub-model 2-20
2.10 Stages 2-21

3 Loads 3-1
3.1 Joint loads 3-2
3.2 Beam loads 3-2
3.3 Element loads 3-2
3.4 Support displacement 3-3
3.5 Combine loads 3-3
3.6 Global loads 3-3
3.7 Solid loads 3-3
3.8 Wind loads 3-3

4 Solve the model 4-1
4.1 Solve - general 4-1
4.2 Solution method 4-2
4.3 Singularity 4-3
4.4 Problems - General 4-4

5 Results 5-1
5.1 Combinations 5-2
5.1.1 Combinations - General 5-2
5.2 Element Coordinate Systems 5-3
5.2.1 Result Coordinate System **X , Y , Z** 5-3
5.2.2 Reinforcement coordinate system 5-5
5.3 Graphic Results 5-6
5.3.1 Results at Element Centres 5-6
5.3.2 Result Contour Map 5-6
5.3.3 Results along a line 5-7
5.3.4 Deflections 5-8
5.4 Tabular results 5-8
5.4.1 Tables - Sign Conventions 5-8
5.4.1.1 Sign Conventions - Beams 5-8
5.4.1.2 Sign Conventions - Quadrilateral and Triangular Elements 5-9
5.4.2 Beams - Axial force, Moments and Shears 5-10
5.4.3 Span Deflections 5-12
5.4.4 Element results 5-12
5.4.5 Reactions 5-13

6 Dynamic analysis 6-1
6.1 Mode Shape and Natural Frequency Analysis 6-1
6.2 Seismic Analysis 6-1
6.3 Forced Vibrations and Time History Response 6-3

7 Structural Steel Design 7-1
7.1 Steel Design - General 7-1
7.2 Steel section tables and selection 7-2
7.3 Creating a Steel Structure from the *STRAP* Model 7-3
7.4 Compute 7-4
7.5 General arrangement drawing 7-4
7.6 Example 7-7

8 Concrete Design 8-1
8.1 General 8-1
8.2 Sections 8-2
8.3 Seismic design 8-2
8.3.1 Beams 8-3
8.3.2 Columns 8-3
8.3.3 Walls 8-4
8.4 Design Procedure 8-5
8.4.1 Non-seismic 8-5
8.4.2 Design procedure - seismic 8-6
8.5 Draw columns 8-7
8.6 Draw slabs 8-8

9 Bridge Postprocessor 9-1
9.1 Introduction 9-1
9.2 Solution Method and Program Assumptions 9-1
9.3 How to use the Bridge Module 9-2

10 POSTTEN 10-1
10.1 General 10-1
10.2 How to use the program 10-1
10.3 How to define cables 10-3
10.4 Magnel diagrams 10-4

11 Foundation Design 11-1
11.1 Program Specifications 11-1
11.2 How to use FOUND 11-2
11.2.1 Stand – alone program 11-2
11.2.2 STRAP post – processor 11-2

12 Section generator - CROSEC 12-1
12.1 Program Specifications 12-1
12.2 How to use the program 12-2
12.3 Hints and suggestions 12-2

A1 Hardware Requirements and Program Capacity A-1

A2 Printing the Manual A-2

*** Blank page ***

1 General

This User's Manual provides a concise summary of the main features of the STRAP structural analysis and design program.

To print the full User's Manual, refer to Appendix A2

Detailed explanations on all of the options and features are available in the on-line Help supplied with the program.

To display the entire User's Manual, select the Help option in the program menu bar

To display Help for the current option, press [F1]

For more information on Help, refer to section 1.9.5

STRAP (STRuctural Analysis Programs) is a package of computer programs for the analysis of linear elastic structural models. The system includes static analysis and dynamic response analysis.

STRAP programs offer the engineer an easy-to-use tool for analysing a wide range of frame and truss structures, continuum mechanics problems (plates, shells, etc.) or a combination of both types.

Frame structures consist of an assemblage of one-dimensional beam elements defined by the cross-section properties. Continuum mechanics problems are solved using the finite element method: the structure is modeled by two- or three-dimensional area elements interconnected only at nodal points. The behaviour of a finite element model closely approximates the behaviour of the real structure.

STRAP includes the following elements:

- One-dimensional : Beam element
- Two dimensional : Quadrilateral element
  Triangular element
- Three-dimensional : Solid element

The program allows elements of all types to be defined in the same model. This is very useful when analyzing structures such as plates with edge beams, combined frames and shear-walls, etc.

The method of solution is the well-known stiffness method. It assumes a linear elastic model and small displacements. The program solves the stiffness equation:

$$[K] * \{d\} = \{P\}$$

where $[K]$ = the structure stiffness matrix
$\{d\}$ = the unknown nodal displacements
$\{P\}$ = the applied nodal forces

After solving the equation for the unknown joint displacements, the program calculates the internal forces and stresses in the beams and elements.

Although using *STRAP* does not require knowledge of computer structural analysis, it is recommended that the user familiarize himself with the basic theories of the stiffness method and finite elements.

The *STRAP* system consists of the following modules:

- **Static Analysis**

  - Plane frame analysis:
    Analysis of plane frame models (all loads applied in the plane of the model) composed of beam elements and two dimensional plane stress elements.

  - Plane grid analysis:
    Analysis of plane grid or plate models (all loads applied perpendicular to the plane of the model) composed of beam elements and two dimensional plate bending elements).

  - Space frame analysis:
    Analysis of general space structures composed of beam elements and combined plane stress/plate bending elements. Typical structures are space frames, shear walls combined with frames, roof shells, folded plates, water tanks, etc.

  - Truss:
    Plane and space truss analysis composed of pinned beam elements only.

- **Dynamic Analysis**

  - Dynamic modal shape analysis and natural frequency calculation.

  - Seismic response spectrum analysis

  - Forced Vibrations and transient Response

- **Design Postprocessors**

  - Structural steel design and member selection according to AISC-LRFD, AISC-ASD, Eurocode 3, BS5950, AASHTO-LRFD/ASD, CAN CSA S16.1, IS:800 and AISI-ASD & LRFD (Cold-formed).

  - Reinforced concrete design for beams and columns according to ACI 318, CSA A23.3, Eurocode 2, BS8110 and IS:456.

  - Bridge design:
    Defines lanes; calculate influence lines and maximum/minimum results for each result type for all points on a bridge model.

  - Foundation design:
    Design of spread footings.

The most general and comprehensive module in the package is "Space Frame". It enables the engineer to model any type of space structure using one and two dimensional elements.

In order to simplify the use of the programs, especially for engineers unfamiliar with computer structural analysis, the system also includes modules for more specific types of structures. For example, the "Plane Frame" module solves plane frames only. "Space frame" can also be used to solve plane frames, but it is more convenient to use "Plane frame" as the data required is simplified and the possibility of user error is reduced accordingly.

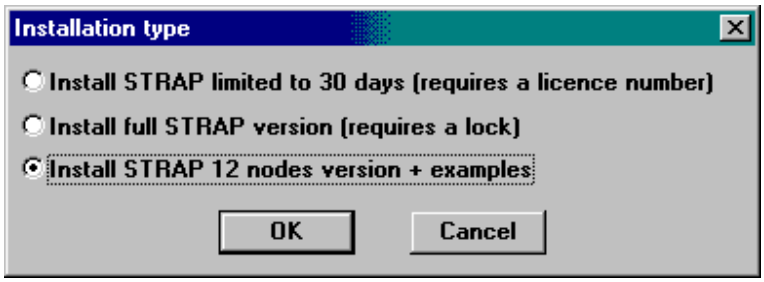
1.1 Installing the Program

Before starting the program installation, make sure that you have the hardware required to run *STRAP*:
- A personal computer with Windows 98/2000/NT/XP
- At least 35 MB of free space on your hard disk

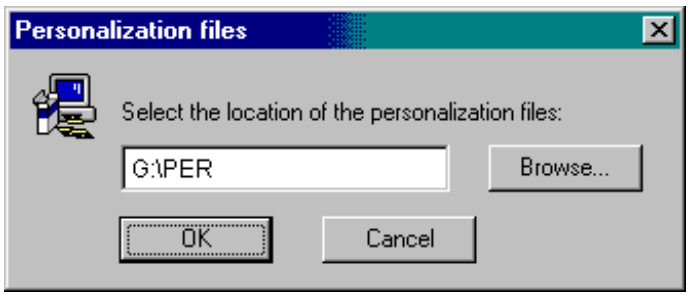
The files on the *STRAP* CD are in compressed format and are copied to the hard disk and expanded by an installation program. The program cannot be installed by simply copying the files from the CD.

To install the program:
- Insert the CD in its drive.
- Follow the instructions on the CD case back cover
- Follow the instructions displayed on the screen:

Three different versions may be installed from the *STRAP* CD:

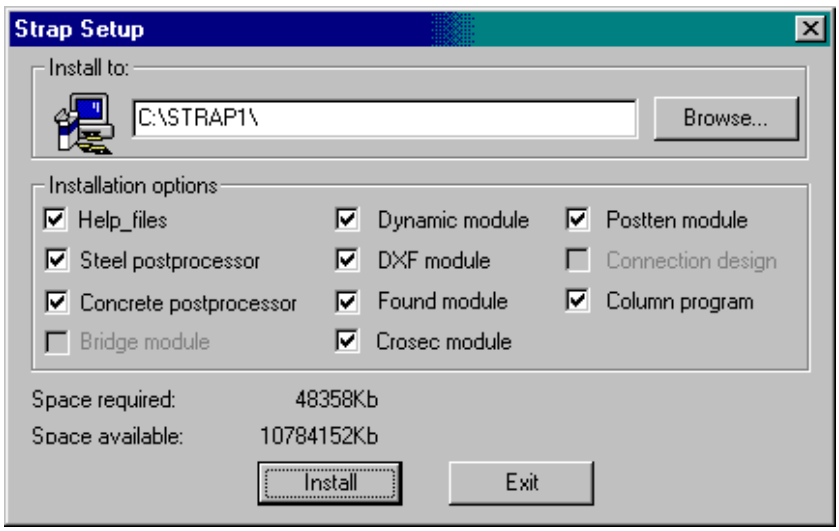

Installation type
Install STRAP limited to 30 days (requires a licence number)
Install full STRAP version (requires a lock)
Install STRAP 12 nodes version + examples
OK
Cancel

If you select **Install full STRAP version**, the program searches for "Personalization files" on the CD (not required for the "12 nodes" or "30 days" versions):



← *Click* **OK**

Then select the program directory and the modules to be installed:



The program should be installed in a **separate** directory (the installation program opens a new directory if the directory with the name entered does not exist).

- To change the program directory, click Browse
- Set the required program modules to ☑, Note that the entire program does not have to be installed, a useful feature for updates.
- Click Install to begin the installation.

For network installation:

- the **HINSTALL** utility program must be run once from every station where *STRAP* will be used. This utility is installed in the *STRAP* program directory and may be run either as a DOS command or using the Windows **Run** option.

  Run the command using the **/I** switch. For example: **G:\HINSTALL\HINSTALL /I**
- Run the **LMSETUP.EXE** program from the \NETHASP directory on the CD program either by entering a DOS command or using the Windows **Run** option. The program must be run from the computer that has the network plug attached to it.

When running *STRAP*:

- If you are using plug-protection (dongle), be sure to insert the plug in the parallel or USB port.
- If you installed the "30 day" version, the program will request license numbers the first time that you run *STRAP*. Contact your *STRAP* dealer.
- The **Setup** option in the initial screen toolbar contains many program configuration options. Refer to 1.4.

1.1.1 Tutorial

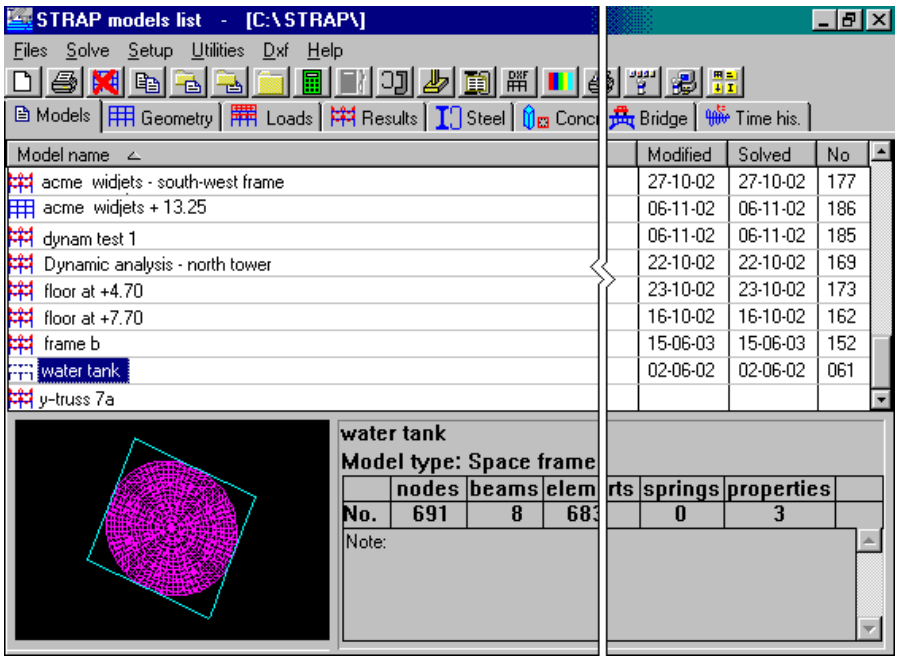
STRAP is supplied with several tutorial models. The model files are automatically copied to folder \STRAP1\DEMO; Specify this folder as the "Current directory" if you want to define/modify the tutorial models. Refer to 1.3.3.

To print the Tutorial User's Manual, refer to Appendix A2.

1.2 Running the program

The program Main Menu is displayed when the ***STRAP*** icon is selected in the Windows screen or when the Models tab is clicked in any of the program modules:

- the program lists the existing models in the current directory, sorted alphabetically, by date or by model number.
- clicking on any line highlights the model title; the program displays the latest view of the model and its statistics at the bottom of the screen.
- clicking on a tab below the toolbar will initiate the relevant option for the highlighted model.
- multiple models may be selected for the delete and copy model options.


STRAP models list - [C:\STRAP\]
Files Solve Setup Utilities Dxf Help
Models Geometry Loads Results Steel Conc Bridge Time his.
Model name
Modified Solved No
acme widjets - south-west frame 27-10-02 27-10-02 177
acme widjets + 13.25 06-11-02 06-11-02 186
dynam test 1 06-11-02 06-11-02 185
Dynamic analysis - north tower 22-10-02 22-10-02 169
floor at +4.70 23-10-02 23-10-02 173
floor at +7.70 16-10-02 16-10-02 162
frame b 15-06-03 15-06-03 152
water tank 02-06-02 02-06-02 061
y-truss 7a
water tank
Model type: Space frame
nodes beams elem rts springs properties
No. 691 8 683 0 3
Note:

The list of model titles is displayed sorted according to any of the four columns in the table:

- model title
- date revised
- date solved
- model number

To sort the models according to one of the columns, click on the column header; a ▲ will be displayed alongside the column header, indicating that the column is sorted in ascending order (e.g. sorted by model title - models starting with 'A' are displayed first; sorted by date - the oldest model is displayed first). Click again to sort in descending order - the symbol in the header will be revised to ▼.

- The small icon displays the current status of the model:

  only geometry has been defined

  geometry and loads have been defined

  the model has been solved (results may be displayed)

  dynamic mode shapes have been calculated (no static loads have been defined)

To create/amend a model:

- Define a new model - refer to 1.2.1
- Amend/rerun an existing model - refer to 1.2.2

The following options are available in the toolbar:

- "Files" pull-down menu - refer to 1.3
- Setup - refer to 1.4.
- Utilities - refer to 1.5.
- DXF import/export options - refer to 1.6.

Note:

- The program stores data for each model in a series of files. All file names for a particular model include a number assigned by the program when the model is created. For example, if the number displayed is "017", then the model files are:
  - **GEOM017.DAT**(geometry)
  - **STAT017.DAT** (loads)
  - **RESLT017.DAT** (results)
  - etc.

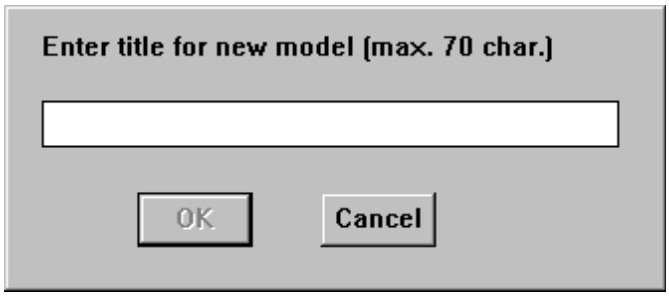
  File management is handled by the program and this number is displayed for information only.
- corrupted model lists may be restored by using the **Files** - **Utilities** - Recreate a models list option. Refer to 1.5.5

1.2.1 Define a new model

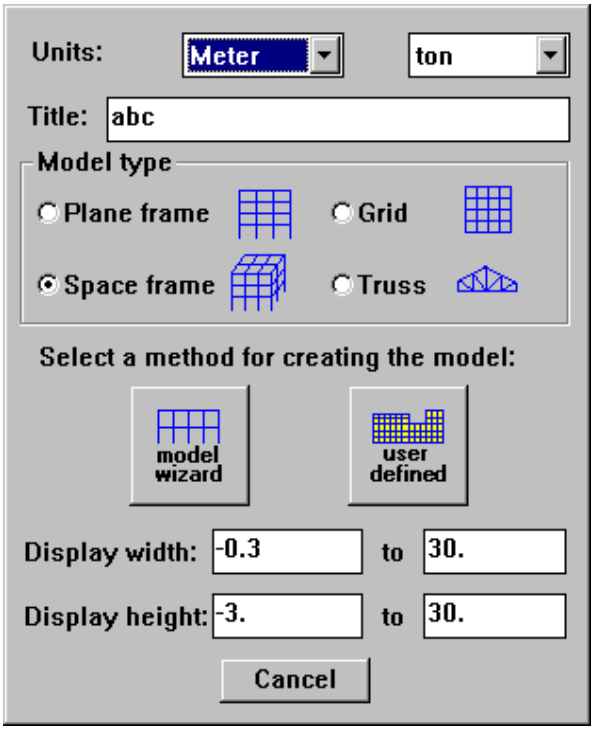
- select **Files** in the menu bar
- select **New model** in the pull-down menu.

  or click the icon in the icon bar.

- define the model title:



The program then displays the Geometry preliminary menu :



← Select the model units

← revise the model title

← Specify the model type

← Select the method for defining the new model:

There are two methods for defining a new model:

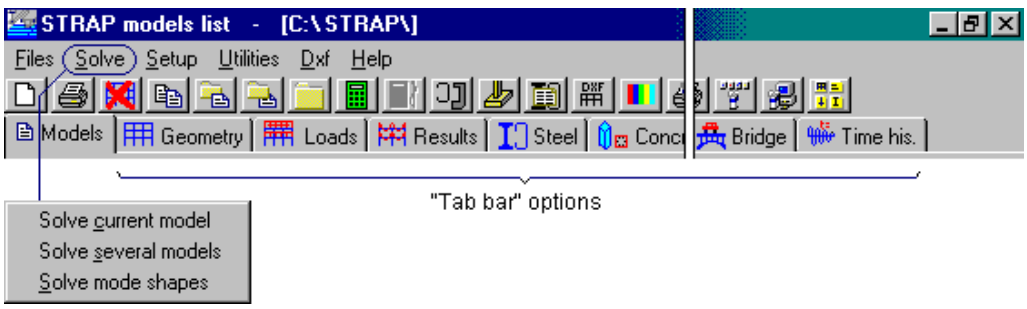
Select a model from the *STRAP* library of standard structure types. Define the geometry, loads and load combinations. The basic model created by the wizard may be revised later using any of the *STRAP* geometry and loading definition options.

The program displays a blank screen. Define the geometry and loads from scratch using any of the *STRAP* geometry and loading definition options.

1.2.2 Amend/rerun an existing model

To revise a model that appears on the list:

- To revise the geometry:
  move the cursor to the model name in the list and double-click the mouse
- To directly access any other part of the program (loads, results, etc):
  - move the cursor to the model name in the list and click the mouse once; the model name will be highlighted.
  - click one of the options in the 'tab bar' below the toolbar:

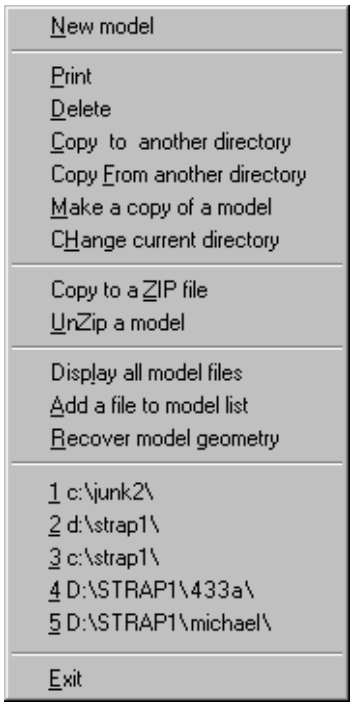


*** Blank page ***

1.3 Main menu - "File" options

Geometry, loads, stiffness matrix, results, etc. are stored in separate files. The program automatically creates and names all of the data files for a model. The user is not required to be familiar with the names and types of files available. For all options, the appropriate model is selected from the model list displayed.

If you select the option **Files** in the menu bar, the following pull-down menu is displayed:



The "File management" operations should always be carried out by using the options in this section and never by using Windows File Manager programs (e.g. "Explorer") or the DOS "DEL", "COPY", etc. commands.

← The last five folders selected in the **Change current directory** option are displayed at the bottom of the of the File menu. Clicking one of these lines will make that directory the new current directory.

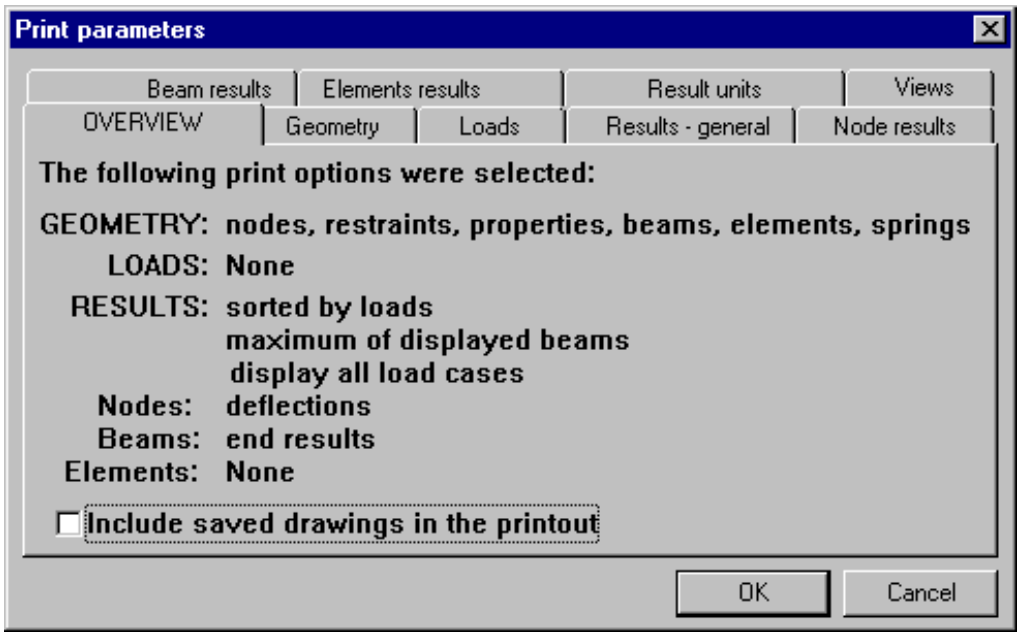
- New model - Refer to 1.2.1.
- Print - Refer to 1.3.1.
- Delete
Copy to another directory
Copy from another directory
Make a copy of a model - Refer to 1.3.2.
- Change current directory - Refer to 1.3.3.
- Copy to a ZIP file - Refer to 1.3.4.
- Unzip a model - Refer to 1.3.5.
- Display all model files - Refer to 1.3.6.
- Add a file to the model list - Refer to 1.3.7.
- Recover model geometry - Refer to 1.3.8

1.3.1 Print

Use this option to print all or selected input, results and saved drawings (in any order) for any model in the list without running the model.

- Specify the tables to be printed by clicking on the **Beam results**, **element results**,...tabs.
- Specify the load cases/combinations to be printed in the **Loads** tab
- Print input/output data for specific elements/nodes only by specifying one or more views in the **Views** tab
- ☑ **Include saved drawings** to add drawings created with the **Save for "print/edit" drawing** option

The current selections are displayed in the **Overview** tab. Click OK to start printing



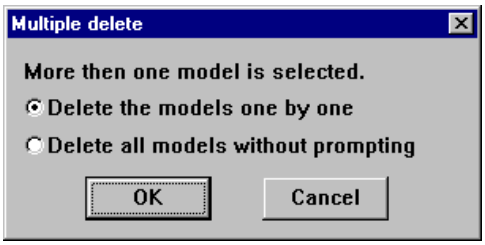
1.3.2 File management

1.3.2.1 Delete

This option deletes an entire model from the list or erases selected files only, e.g. loads, results, etc.. Note that multiple models may be selected.

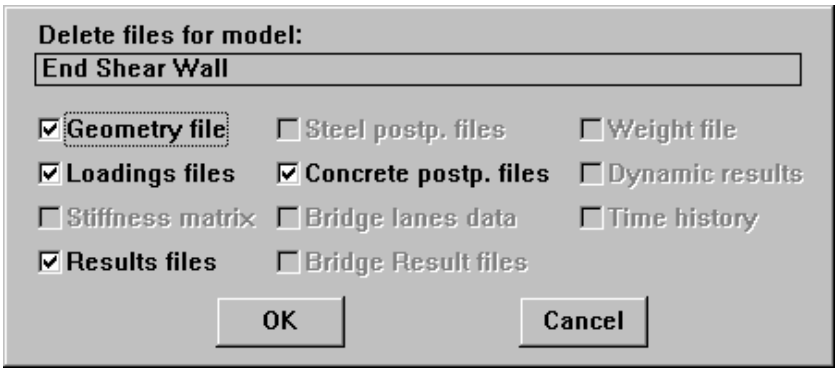
To delete models or files:

- click and highlight model names in the list; to select more than one model, press the [Ctrl] key when clicking on the additional models
- if you selected multiple models:



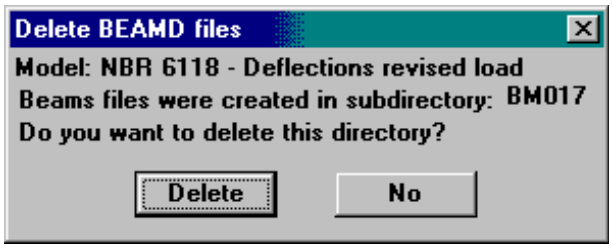
- ◉ **Delete models one-by-one**
  the following menu will be displayed for each model selected
- ◉ **Delete all models without prompting**
  the following menu will be displayed only once and the selection will be used for all models

- select the files to be deleted:



- only files that are marked with a ☑ will be copied. Click on the file description to toggle the status.
- click the OK button.

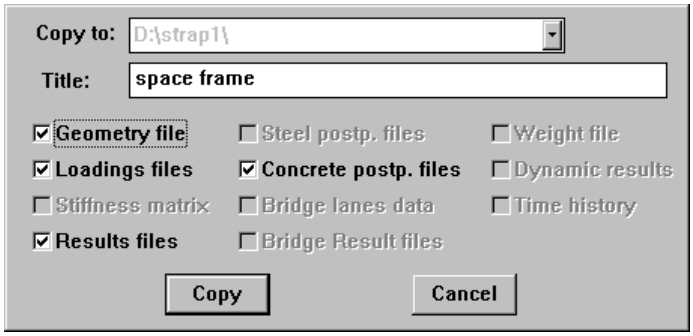
Note:
- only the relevant files will be displayed in the menu
- if you delete geometry files, the program will delete the entire model.
- if you delete loads, the program will also delete all result files.
- If BEAMD files were created for the model, the files may also be deleted:


Delete BEAMD files
Model: NBR 6118 - Deflections revised load
Beams files were created in subdirectory: BM017
Do you want to delete this directory?
Delete
No

To delete a model from a floppy diskette:
- select **Change current directory** and specify A:
- proceed as explained above.

1.3.2.2 Make a copy of a model

Use this option to create a copy of the highlighted model in the current directory:

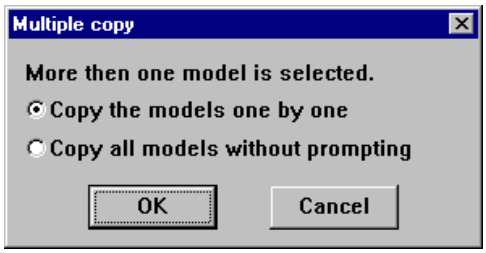


- Revise the **Title** (optional).
- only files that are marked with a ☑ will be copied. Click on the file description to toggle the status.
- press the Copy button to begin copying.

1.3.2.3 Copy to another directory

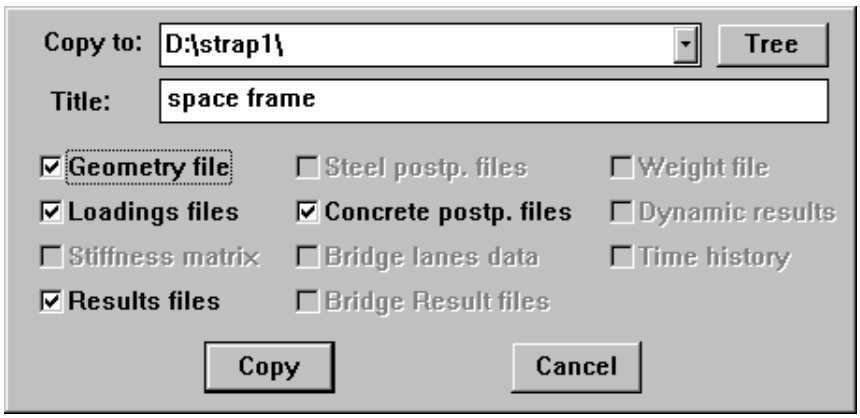
Use these options to copy a model to another directory, another directory or to backup a model to a floppy diskette. Note that multiple models may be selected:

To copy models:
- click and highlight model names in the list; to select more than one model, press the [Ctrl] key when clicking on the additional models
- if you selected multiple models:



Copy models one-by-one
the following menu will be displayed for each model selected
Copy all models without prompting
the following menu will be displayed only once and the selection will be used for all models

- select the files to be copied:



- type the drive/directory in the **Copy to** box, select a directory using the Tree option, or click the ▼ to choose a recently selected directory; press [Enter] or click the Select model button.
- Revise the **Title** (optional).
- only files that are marked with a ☑ will be copied. Click on the file description to toggle the status.
- press the Copy button to begin copying.

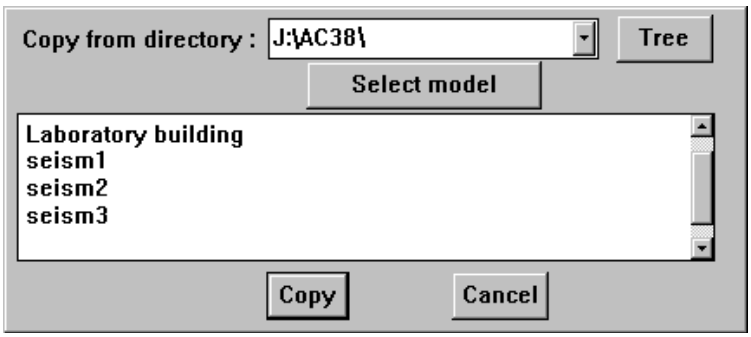
Note:
- when copying multiple or large models to a diskette, the program displays a message with the number of additional diskettes required and notifies when to change diskettes. All diskettes must be formatted prior to the backup.
- To create a copy of a model in the Current directory, refer to 1.3.2.2 - Make a copy of a model

1.3.2.4 Copy from another directory

Use this option to copy a model from another directory or to restore a model from a floppy diskette. Note that multiple models may be selected.

To copy the models:

- type the drive/directory in the **Copy from directory** box, select a directory using the Tree option, or click the to choose a recently selected directory; press [Enter] or click the Select model button.
- select the model or models to be copied:

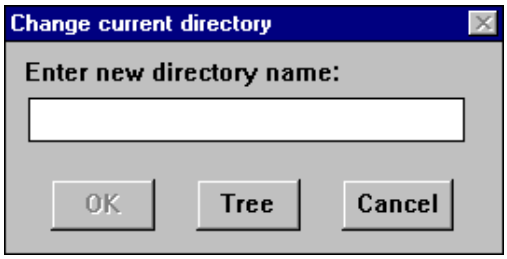


  to select more than one model, press the [Ctrl] key when clicking on the additional models
- for each model selected, specify the files to be copied from the list displayed; only files that are marked with a will be copied. Click on the file description to toggle the status.
- click the Copy button to begin copying.

Note:
- if the model was copied to two or more diskettes, the program will instruct you change diskettes.
- to create a copy of a model in the current directory, refer to Make a copy of a model. (1.3.2.2)

1.3.3 Change current directory

The current directory is displayed at the top of the screen.



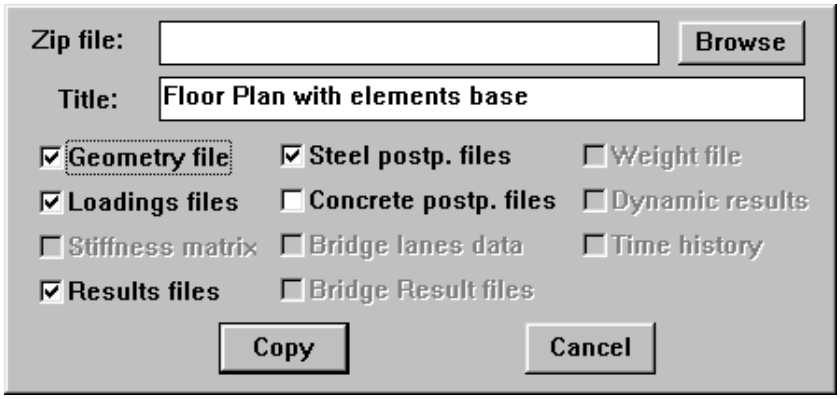
- to select an existing disk directory on any drive, type in the name of the path or click the **Tree** button and select the path in the standard Windows "**Select a directory**" dialog box.
- To create a new directory, click the **Tree** button and define a new path by clicking the icon ("Create New Folder")in the standard Windows "**Select a directory**" dialog box.

Note:
- the last five directories selected in this option are displayed at the bottom of the File menu and may be selected by clicking on them
- do not select a “Read-only” directory

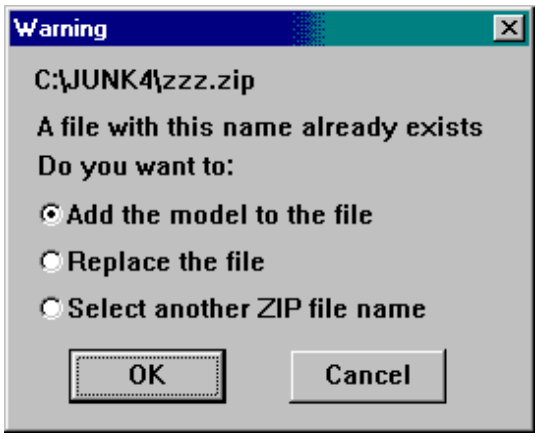
1.3.4 Copy to a ZIP file

Model files may be condensed to a single ZIP format file and may be imported back into the model list from a ZIP file.

To copy to a ZIP file:
- enter the name of a new ZIP file or select an existing one; specify which of the data files to condense:



- for an existing ZIP file, select one of the following options:

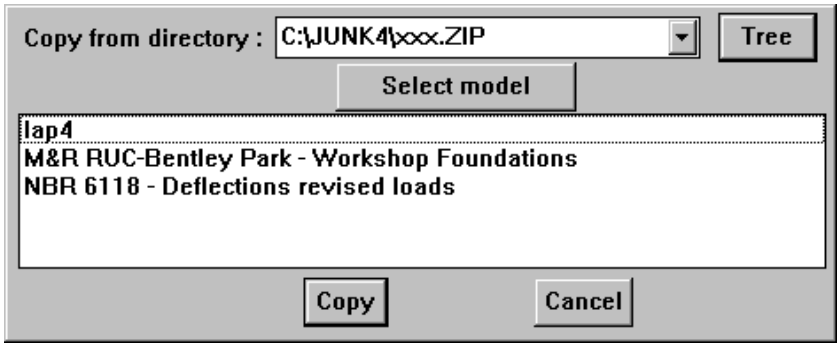


where:
- **Add the model**
  the selected models are added to the file
- **Replace the file**
  the exiting file is erased and a new one is created with the selected models
- **Select another ZIP file name**
  Cancel the option and enter a different file name before continuing.

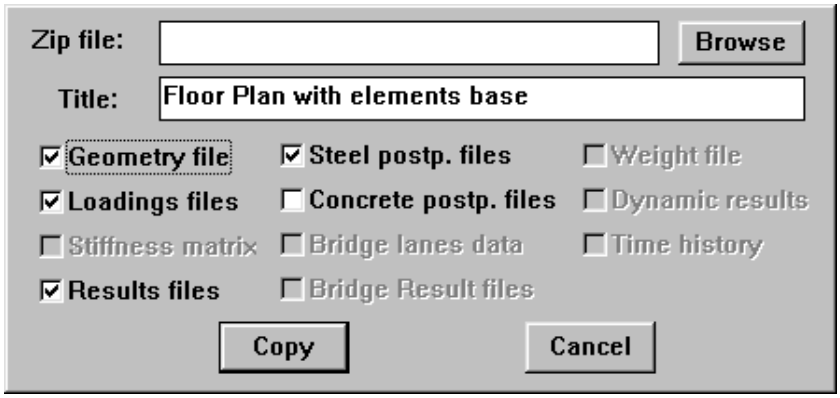
1.3.5 UnZIP a model

Models may be retrieved from ZIP files; note that the ZIP files do not have to be created by *STRAP* but they must contain the directory file ZZMODEL.DIR.

To retrieve a model from a ZIP file:
- select the ZIP file
- click and highlight one or more of the models in the list:



- click **Copy**
- select the files to extract:



- click **Copy**.

1.3.6 Display model file names

The program automatically assigns file names to all files created for a model.

Use this option to display the file names for the highlighted model.

1.3.7 Add a file to model list

The model list is maintained by the program and displays all models in the current directory, both those created by the user by running *STRAP* in the directory or those copied to the directory using the program "Copy to" and "Copy from" options. Note that the model list is stored in the file ZZMODEL.DIR; this file is found in every working directory and backup diskette.

Models whose files are manually copied (e.g. using DOS "Copy" command) to the current directory will not be displayed in the model list because the ZZMODEL.DIR file cannot be edited by the user.

This situation may occur in several ways. For example:
- the user defines the model by writing a geometry ASCII file and a loading ASCII file
- the GEOMnnn.DAT file is corrupted and the user wants to recreate the model from the GEOINnnn.DAT file (a list of geometry commands maintained by the program when the user defines the model)
- the model files are available (e.g. on a backup diskette) but the ZZMODEL.DIR file is not present or corrupted; the program "Copy to" and "Copy from" options will not locate the models.

Use this option to add such models to the list.

Note:
- the files may have any name
- the geometry file must start with the command REPLACE or ADD (refer to 1.11.1 – Batch mode - geometry).
- the loading file must start with the command ASCII (refer to 1.11.2 – Batch mode - loads)

The geometry file may be in one of two formats:

ASCII (STRAP batch mode commands)
the user created a geometry ASCII file (refer to 1.11.1 - Batch mode-geometry for more details on file format).

STRAP internal format
the geometry file was created by *STRAP* , i.e. binary format. This type of file cannot be edited by the user.

Type in the name of the geometry and loading file (optional).

Note:
- a drive name (e.g. A:) or a directory path may be entered.
- click the button to display a list of files in the current directory.

1.3.8 Recover model geometry

The current geometry for each model is stored in a binary file named **GEOMnnn.DAT**, (where "nnn" indicates the model number). However, ***STRAP*** simultaneously creates an ASCII data file for each model named **GEOINnnn.DAT** that contains the geometry data in the form of commands. When the *STRAP* geometry file is corrupted, the program may be able to recreate all or part of the file by reading the commands in the GEOINnnn.DAT file.

Always backup the model before recovering the geometry. In the event that the Recover is not successful, please contact your *STRAP* dealer.

1.4 Setup

Use this option to specify default values for colours, units, standard material properties, output format, etc.

Default values for *STRAP* program parameters are initially stored in the file **STRAP.INI,** located in the program directory. Note that there is only one STRAP.INI file for all users.

When a user revises a Setup parameter, the program writes the new value in the Windows "Registry". Each user is allocated a separate section in the Registry, so each user can create his own personal set of Setup parameters. Note that the *STRAP* installation program does not write any Setup values in the "Registry"

The program initially searches for default values in the Registry; if it does not find a value (i.e. the user has not revised the parameter in Setup) it takes the default value from STRAP.INI.

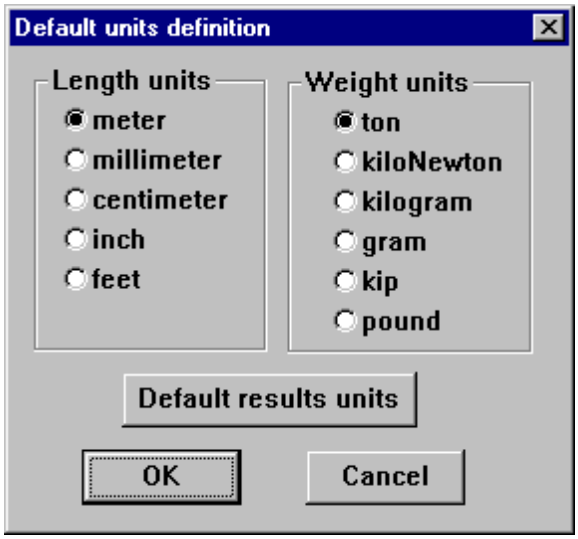
1.4.1 Setup – Colors

You may specify the permanent screen colors for:
- background
- graphic lines, e.g. beams, elements, various loads, etc.
- text associated with each graphic line
- colors for contour map fills

1.4.2 Miscellaneous

Units

Set the default input and result force and length units for all new models.



Set the default input and result units for **all** new models. (if the default units are revised at the start of the geometry definition, the revision will apply only for that particular model).

Materials

The properties of 10 materials are permanently stored in the program. Four of these materials are user-defined materials. These 10 properties will be displayed when the **Materials** option is selected in beam or element property definition.

The properties of all 10 materials may be edited.

The properties are:
- Modulus of elasticity
- Poisson's ratio
- Density
- Thermal expansion coefficient (1/°Celsius or 1/°Fahrenheit)

Units:
- Specify the units for modulus-of-elasticity and density in the list boxes at the top of the screen.
- The thermal coefficient may be defined according to either unit. However the temperature difference value entered when a temperature load is applied must be according to the same units.

Note:
- maximum name length = 4 characters
- properties may be defined in exponential format
- additional user-defined materials may be defined in each model **for that particular model only**.
- revising the program materials will not affect the material properties of existing models.

Miscellaneous

Refer to the program Help or the full manual.

Rendering

Refer to the program Help or the full manual.

1.4.3 Print parameters

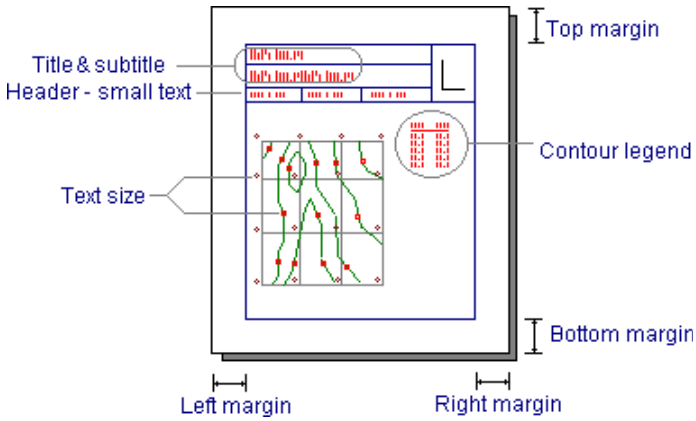
Table print styles

The format for ***STRAP*** tabular output may be specified by the user.

A series of "styles" may be defined using this option; each style contains information on the fonts, margins, lines and spacing to be used when printing the table. When printing tables when in any geometry, loading, result or postprocessor modules, the user may select one of the predefined styles and the tables will be printed in the specified format.

Print drawing

Specify the following items:



Note:
- the margins are relative to the maximum print area available for your printer (i.e. margins may be present when zero values are entered in the above menu). Refer to your printer manual.

Define the width of lines associated with each drawing element.

Note:
- the line widths are defined in millimeters.
- a line defined with zero width will be drawn with a width of one dot, e.g. for a printer resolution of 300 dpi, the line width will be 1/300 inch.
- geometry line widths also apply to the geometry for loads and results.

1.4.4 Toolbars

Icon bars are displayed above the graphic display during geometry and load definition. Clicking an icon on the bar operates the option that it represents directly. For example:

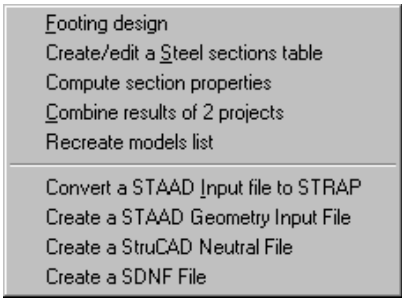
Use this option to customize the icon bars.

Up to 10 different icon bars may be defined (7 "default" bars are present when the program is installed). The program automatically displays icon bar no.1 on the screen Clicking the icons at the end of the bar displays the following/previous bars.

** Blank page **

1.5 Utilities

The following utility modules are available:


Footing design
Create/edit a Steel sections table
Compute section properties
Combine results of 2 projects
Recreate models list
Convert a STAAD Input file to STRAP
Create a STAAD Geometry Input File
Create a StruCAD Neutral File
Create a SDNF File

Note that you can add options to this menu. This is convenient if you want to run a *STRAP* batch operation according to the *STBatch* utility, but any Windows "Target" command can be initiated (i.e. run any program).

The *STBatch* utility can be used for running *STRAP* in batch mode or for generating ASCII files with geometry, load and/or result data in user specified format. Refer to the *STBatch* manual or *STRAP* Help for more details.

1.5.1 Footing design

Design reinforced concrete spread footings. Refer to 11.

1.5.2 User steel section table

This utility module allows you to:
- build a customized "user" steel section table for the *STRAP* geometry and the Steel postprocessor. Sections may be copied from any of the standard steel tables (British, European or American) or new sections may be defined by entering their dimensions; the program will calculate all the section constants.
- add cold-formed sections to the standard steel tables.

Note:
- the "user" steel table is always stored in PROPTABS.DAT and must be in the program directory.
- note that new section types may also be defined for convenience.
- if a user steel table has been created, **The user steel table** option is displayed in the Section table menu.

1.5.3 Compute section properties

A module that calculates the properties of geometric sections (area, moment-of-inertia, center-of-gravity, etc). Also available as a stand-alone program - CROSEC. Refer to 12.

The program can calculate all properties that are required for the design of cold-formed (light gauge) sections. All line sections (with any arbitrary shape) defined in this program may be transferred to the *STRAP* steel postprocessor and can be calculated as cold-formed sections.

Two general types of sections may be defined:
- Line sections: Sections composed of a series of connected lines, each with a specified thickness.
  - Segments may be defined in any direction (diagonal).
  - Several 'subsections' may be defined to form a section.
  - The properties may be copied to *STRAP* geometry
  - Tables of cold-formed sections may be added to the *STRAP* property tables
  - Properties calculated include: Shear center, warping constant, torsional moment-of-inertia and torsional-flexural buckling constant
- Solid sections: Sections formed by a closed contour.
  - Several separate contours may be defined to form a section; additional contours may be "holes".
  - The properties may be copied to *STRAP* geometry.
  - Properties calculated include: Torsional moment-of-inertia (exact)

1.5.4 Combine results of two projects

Use this utility to combine the result files from several models. This option is essential in models where the geometry changes for different loading cases (e.g. supports, properties, etc.) and the structure must be run as two or more separate models, but where it is necessary to search all loading cases from all of the models in the various design post-processors.

**** the Stage option (2.10) can achieve the same result more efficiently ****

The number of nodes or elements in the two models that are combined do not have to be identical but the node/element at the same location in both models must have the identical numbering.

By combining STATnnn.DAT, the program adds the load commands from the load cases of the second model to those of the first. If you later decide to rerun the first model, remember to erase the unnecessary load cases (the load commands are required by the various post-processors to calculate the span moment diagrams).

The process may be repeated to add additional models to the combined ones; the models must be added in order of decreasing size (largest to smallest).

Note: **Only the first model selected is revised**

1.5.5 Recreate the model list

Use this option to recreate a corrupted model list. The model list is stored in the file **ZZMODEL.DIR** (each working directory has a different ZZMODEL.DIR file). This option scans the current directory to locate all model files (GEOMnnn.DAT, STATnnn.DAT, etc) and rebuilds the model list.

Note:
- the new model list file is named **ZZMODEL.NEW**. The corrupted ZZMODEL.DIR file is not erased.
- to use the new file, copy it to ZZMODEL.DIR.

1.5.6 STAAD File Conversion

This utility module converts STAAD input files to *STRAP* format. Geometry, loading and load combination commands are translated.
- select **Files** in the Menu bar on the ***STRAP*** initial screen
- select **Utilities** in the pull-down menu.
- select **Convert a STAAD input file to STRAP.**
- type in the name of the STAAD file

Note:
- The program creates the ***STRAP*** GEOMnnn.DAT, STATnnn.DAT, and COMBnnn.DAT files.
- STAAD commands with syntax errors are not translated. Error messages are listed on the screen and written in an ASCII file ERRS.LST.
- The program does not check that entire geometry has been defined (e.g. restraints).

1.5.7 Create a STAAD geometry input file

Create STAAD format geometry and load files for the current model

1.5.8 Create steel detailing files

Create a structural steel detailing file in one of the following formats:
- STRUCAD
- SDNF

The program writes the model geometry and the beam end results to the files. The steel section written to the file is the current one in STRAP geometry; different sections selected in the Steel Postprocessor must be transferred to the geometry using the **Exit and update geometry** option in the postprocessor.

1.6 DXF options

1.6.1 Convert metafile to DXF

Use this option to create a two-dimensional DXF file of any ***STRAP*** graphic display.

The metafile is created by selecting **Print drawing** in the **Output** pull-down menu (on the menu bar) and then selecting **Metafile** in the **Send output to:** box.

When this option is selected, the program displays a list of the Metafiles (extensions .EMF, .WMF) in the current directory. Select the file to convert.

1.6.2 Convert STRAP file to DXF (DXF export)

This module converts the ***STRAP*** geometry file to three-dimensional DXF files.

This option differs from the **STRAP Print drawing** option (and subsequent **Convert Metafile to DXF**) in that it creates a full 3D model instead of a 2D image of the current screen display.

- the program works on the current ***STRAP*** model
- plane models generate a 2D DXF drawing
  space models generate a 3D DXF drawing
- Beams generate a DXF "LINE" command.
  Elements generate a DXF "3DFACE" command.
- Each ***STRAP*** property is assigned to a different layer and drawn with a different color in the DXF file.
- Dummy beams and elements are drawn in a separate layer.
- The following is not transferred to the DXF file:
  Dimension lines, sections, materials, text (numbering, etc.).

1.6.3 Create a new model from DXF file (DXF Import)

This module converts a DXF format file to a ***STRAP*** geometry file. It identifies the LINE and mesh commands and converts them to beams or elements, as requested by the user.

Note:
- that each AutoCAD "layer" may be specified as containing beams or elements.
- *STRAP* property group numbers may be assigned to the generated beams/elements according to AutoCAD layer.
- *STRAP* property group numbers may be assigned to the generated beams/elements according to AutoCAD line color

The program handles the different AutoCAD entities as follows:

- Beam layer:
  - each line in AutoCAD is converted to a ***STRAP*** beam.
  - each AutoCAD mesh element (3DFACE) is exploded to a series of lines which are converted to beams.
- Element layer:
  - areas enclosed by three or four lines and creates triangular or quad finite elements.
  - each 3DFACE or surface of a mesh generates a single ***STRAP*** quad or triangular element.

The program also handles intersections between lines, between elements and between lines and elements.

Note:

- DXF space models create ***STRAP*** space models.
  DXF plane models (all Z coordinates = 0) create ***STRAP*** plane frames.
- The program defines element local axes according to the ***STRAP*** defaults.
- The program creates quad finite elements wherever possible; non-planar quads are divided into two triangles.
- Each "3DFACE" element, polygon mesh or polyface mesh surface generates one ***STRAP*** finite element.
- Double lines are treated as single lines.
- Concave elements defined by lines, 3DFACE, polygon mesh or polyface mesh are ignored.
- When creating finite elements the program accepts only quads or triangles. Nodes will be created at intersection points, even if no elements are generated:

1.7 Getting Started

This section is intended mainly for engineers who have no previous experience in the use of computers for structural analysis or who are unfamiliar with the finite element method.

The engineer must prepare a computer model of the structure. The model consists of a series of elements joined at points called nodes (or joints).

STRAP uses two types of elements:

- Beam elements :

  Beams are one dimensional elements that are used to model structural members that can be modeled by a line - beams, columns, bars, etc.

  Beam elements always give accurate results, i.e., if a model naturally consisting of beam elements (e.g., a skeletal frame structure) is analyzed by ***STRAP***, the results will be identical to those calculated manually using any exact method. Similarly, the method of creating the ***STRAP*** model for such structures is usually obvious - each beam or column is represented by a single beam element.

- Finite Elements:

  Finite elements are two-dimensional elements that are used to model surfaces such as plates, shells and walls. They may be either triangular or quadrilateral in shape.

  Finite elements, on the other hand, give inherently inaccurate results (the degree of inaccuracy is usually acceptable by most engineering standards), for the following reasons:

  - The elements should be connected along their common boundaries, but in the finite element model they are connected only at their common nodes. Thus there is a relaxation of continuity along the boundary (although the mathematical development of the elements ensures satisfaction of some, but not all, of the boundary continuity requirements).
  - The mathematical development of the elements assumes a linear stress distribution through the element. In reality, the distribution is usually more parabolic.

  In a continuum structure such as a flat plate there is no natural subdivision of elements, so the structure has to be artificially divided. It is obvious that as the continuum is divided into a finer mesh (smaller elements), the degree of discontinuity is lessened, the stress distribution approaches linearity and the overall accuracy of the solution improves.

  On the other hand, increasing the number of elements in the model increases the solution time and increases the size of the files required to store the input data and results.

1.7.1 Selection of the Computer Model

The preparation of the model for two-dimensional frames, grids or trusses is usually very simple, as each beam and column is represented by a single line element.

More complicated structures - space frames or structures with area elements - require more thought and good engineering judgment in the preparation of the model. Typically, structural members can be represented by either area elements or by line elements and the engineer must choose between them based on his experience.

Remember:

***STRAP* calculates *numerically* accurate results for the defined model.**
It is the engineer's responsibility to:
- **define a model that correctly represents the structure**
- **thoroughly check for illogical or inconsistent results.**

Example 1:

A square concrete plate of uniform thickness, simply supported on all edges. The plate should obviously be modeled by a regular pattern of quadrilateral elements. How many elements are required for a sufficiently accurate solution?

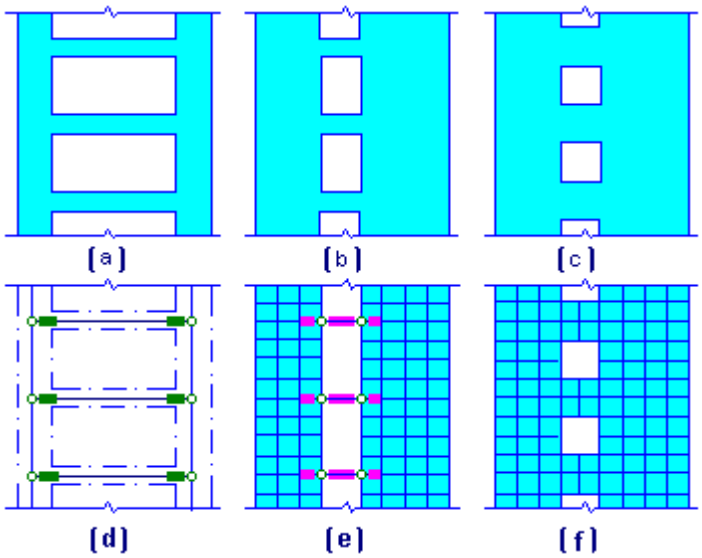
The following table gives results for the center deflection of the plate : (10x10, 0.2 thick, concrete):

	Vertical deflection	
No. of elements		% error
100	(10x10)	0.30
64	(8x8)	0.50
36	(6x6)	0.90
16	(4x4)	2.00
4	(2x2)	6.20

There is no obvious advantage gained from defining the model with more than 36 elements.

Example 2:

Consider the three shear wall structures in Figures (a) to (c). The walls are identical except for the size of the openings.



Figures (d) to (f) show three possible computer models for the corresponding shear walls (a rough mesh has been used for clarity):

- model (a)has relatively narrow walls and beams and so can be modeled entirely by beam elements as shown in (d). Note the 'rigid offsets' in the lintel beams.
- model (b) has wide walls and shallow beams and so can be modeled by the combination of finite elements and beam elements as shown in (e). The lintel beams are extended into the wall for continuity.
- model (c)has wide walls and deep beams and should be modeled entirely by finite elements as shown in (f).

It is important to stress the following points:

- **all** of the models are inaccurate to some degree : Model (a) uses beam elements to model the wall. Model (b) and (c) use finite elements and so the degree of accuracy is dependent on the number of elements used (see below).
- there is no sharp boundary between beams that are considered 'deep' or 'shallow', i.e., it will not always be obvious which model is the most appropriate.
- As opposed to example (a), there are no exact solutions available that the computer results can be compared to. Therefore, good engineering judgment is required to select the correct model.

In summary:

- the model selected must represent the actual structural behavior of the structure.
- accuracy increases as the number of elements increases but both solution time and file size also increase.

Thus, the selection of the model and the division of the model into elements is often a tradeoff between:

- more exact modeling of the structure vs. general simplicity of the model
- increased accuracy vs. reduced solution time and file size.

The following are guidelines for constructing an accurate finite element model:

- **Use quadrilateral elements**:
  In general, try to use quadrilateral elements instead of triangular elements as they give more accurate results.

  Remember that the four corners of a quadrilateral element should all lie on the same plane. If this is not possible, use two triangular elements in place of each quadrilateral.

- **Element shape**:
  - Quadrilateral Elements:
    The greatest accuracy is achieved with a square - 1:1 - element. Elements with a base/height ratio up to 1:2 give good results, but elements with a ratio of 1:5 will be unreliable.

    Try to use rectangular shaped elements whenever possible, If not, the internal angles should not vary greatly from 90°. Angles of 30° or 150° will greatly reduce accuracy. Elements with convex angles should never be used.

  - Triangular Elements:
    Equilateral triangles will produce the most accurate results.

- **Mesh Density**:
  The mesh density need not be constant throughout the model. The program assumes a linear result distribution through the element. If the actual result through the elements is not linear but parabolic, for example, it is obvious that there will be a decrease in the accuracy. In a fine mesh, the result diagram through any one element will always be approximately linear.

  Increase the number of elements where there is a greater rate of change in the internal forces. For example, around supports (where bending moments increase sharply), openings and large concentrated loads.

  To decrease the number of elements:

  Use a rough mesh in areas where relatively low results are expected. Remember that the connection to adjacent elements is through the element end nodes only and so nodes located along an edge of an element between end nodes are ineffective. Use triangular or trapezoidal shaped elements to step between rough and fine quadrilateral meshes.

  If you have doubts as to the accuracy of the results in a particular area of the model, rerun the problem with a finer mesh in that area and compare results. The results converge to the exact solution as the mesh becomes more refined.

- **Models with axes of symmetry**:
  Large symmetric structures can be modeled by defining only one half (or a quarter) of the model. Note that the symmetry must be present in **both** the geometry and loading diagrams.

  Care must be taken to define the correct boundary restraints along the line of symmetry. An error in defining the proper restraints will lead to incorrect results.

1.7.2 Inputting the Model

The program continuously displays the model graphically and updates the display automatically after every input command, thus enabling the user to instantaneously check the accuracy of the input.

STRAP allows you to use one of three input modes:

- **Graphic Mode**:

  The model is defined by moving the crosshair using the mouse/arrow keys to identify node locations, define elements, assign properties, loads, etc. All parameters such as current crosshair coordinate are displayed at the bottom of the screen and are updated continuously. Only a limited number of parameters need be typed in.

  Nodes and elements are numbered automatically by the program when generated. Numbering is always consecutive, unless specified otherwise. The numbering may be revised by the user. The final numbering pattern should be one that will cause the results to be printed in a logical sequence.

- **Command Mode**:

  The model is defined by typing commands in standard format. The geometry is displayed simultaneously on the screen and is updated after every command.

  The numbers of all nodes and elements are specified by the user. The system of numbering should be such that a minimum number of input commands are required. This varies from structure to structure, and is learned mainly from experience. In general, an ordered numbering pattern is recommended.

  When numbering the model, note the following points:
  - The program contains powerful commands for generating the definition of large groups of nodes and elements; these commands require an orderly numbering pattern.
  - Numbering does not have to be consecutive.
  - Nodes which are not connected to the model are ignored.
  - Nodes may be located anywhere along a beam element.
  - Node numbering and element numbering are independent. The structure may contain nodes 1,2,3,4.... as well as elements 1,2,3,4....
  - If a structure consists of both beam and area elements, the same element number may not be given to an element of each type.
  - Solution time is not dependent on the numbering.

  In all cases, it is recommended that the user prepare a sketch showing all numbering before defining the model.

- **Batch Mode**:
  Similar to Command Mode, except that the data is entered in a data file external to ***STRAP*** using any editor program.

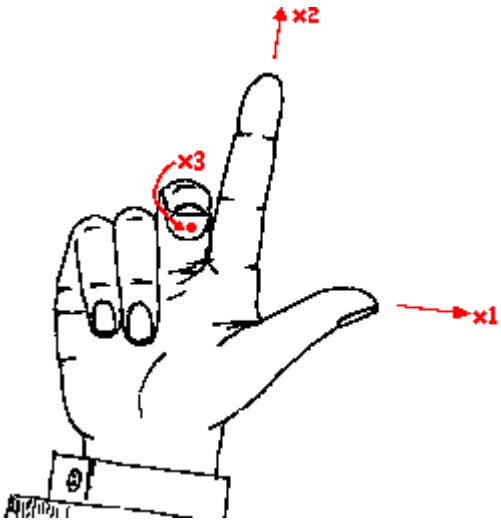
1.8 Coordinate systems

Coordinate reference frames are required to uniquely describe the position of a structure in space, the direction of applied loads and the direction of computed reactions, displacements, forces etc. In addition, the coordinate systems are required to reference other structural information such as element properties.

STRAP uses Cartesian reference frames. A cylindrical system may be used for node coordinate definition.

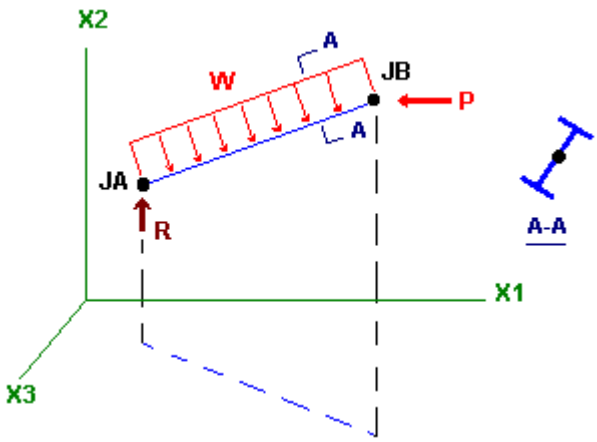
Two types of reference systems are used in ***STRAP***. They are:
- the global coordinate system, denoted by **X1, X2, X3** (uppercase).
- the local element coordinate system, denoted by **x1, x2, x3** (lowercase).

STRAP uses right-handed, Cartesian reference frames.



The X1, X2 and X3 axes (or x1, x2 and x3 axes) are perpendicular to each other and the positive direction of the axes are specified by the 'right-hand rule'. It is obvious that if the directions of any two of the axes are known, then the direction of the third axis is easily determined.

In the following figure a beam element is located in space between nodes JA and JB. The location of the nodes are defined according to the **global** coordinates, i.e. their coordinates relative to the global axes X1,X2,X3.



It is apparent the horizontal load **P** at node JB and the support reaction **R** at node JA should be referenced to the global coordinate system. But it is difficult to define the distributed load **w** on the beam relative to the global axes and obviously a method is required to define the section orientation. It is also apparent that results such as bending moments must be referenced to different directions for each beam.

Consequently, the beam is provided with a **local** axis system x1,x2,x3; section orientation, direction of loads and results are always relative to this local system.

Note that **each** element has its own unique local system independent of the local systems of the other elements in the model.

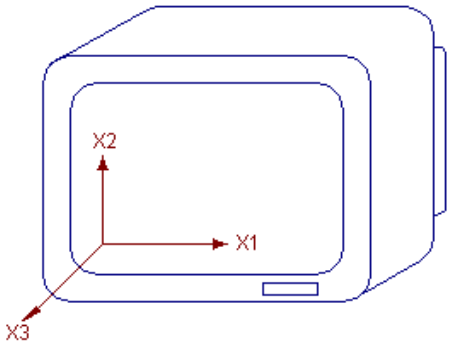
The X1, X2 and X3 axes (or x1, x2 and x3 axes) are always perpendicular to each other and the positive direction of the axes are specified by the right-hand rule. It is obvious that if the directions of any two of the axes are known, then the direction of the third axis is easily determined.

The directions of loads, forces, moments and stresses are referenced to the global or local axes by standard sign conventions.

- Global coordinate system- refer to 1.8.1
- Local coordinate system - refer to 1.8.2
- Sign conventions - refer to 1.8.3

1.8.1 Global Coordinate System

The geometry of the structure, joint loads and displacements, reactions and certain loads are referenced to the global coordinate system. The orientation of the structure with respect to the system is arbitrary and is implied by the engineer by node coordinate input. Generally, one or more global axes are selected to be parallel to one or more characteristic directions of the structure.



The default convention is:
- X1 = horizontal axis on screen.
- X2 = vertical axis on screen.
- X3 = axis perpendicular to the screen and pointing towards the user.

Plane structures are always defined on the X1-X2 plane.

1.8.2 Local Coordinate System

Each element of a structure modeled by ***STRAP*** has a local reference frame associated with it. Element properties, certain loads, member end and internal forces are referred to this local coordinate system.

The local axes directions are automatically specified by the program according to default conventions when the elements are defined, but may be revised by the user.
- Beam elements - refer to 1.8.2.1
- Quadrilateral finite elements - refer to 1.8.2.2
- Triangular finite elements - refer to 1.8.2.3
- Wall elements - refer to 1.8.2.4

1.8.2.1 Beam Elements

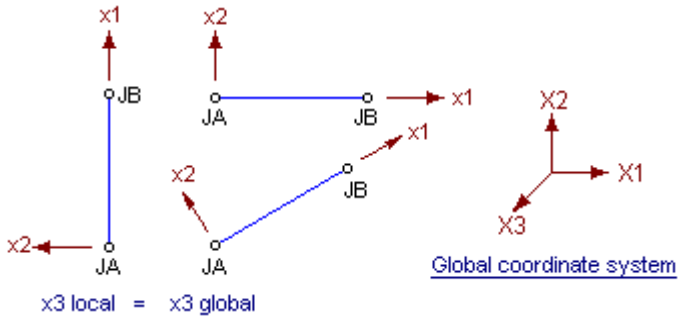
The directions of the local axes determine:
- the orientation of the sections major and minor axes, which are **always** aligned with the x2, x3 axes.
- the direction of beam loads, which are **always** parallel to the x2, x3 axes.

Default Convention:

Plane models:

Axis	Direction
x1	**Always** coincides with the centroidal axis of the beam; the positive direction is from the start node (JA in the geometry tables) to the end node (JB), as specified during the definition of the beam.
x2	As x1 and x3 are known, x2 is determined from the right-hand rule
x3	**Always** parallel to the global X3 axis

Examples:

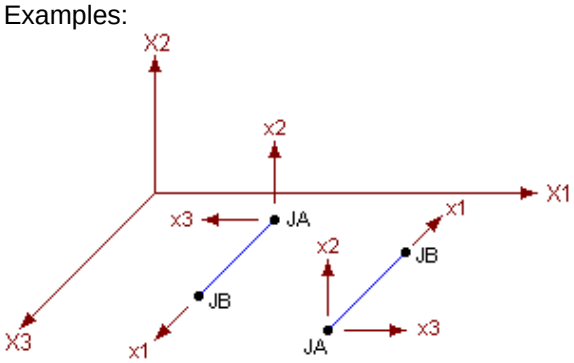


Space models:

Axis	Direction	
	General case: x1 **not** parallel to X3	Special case: x1 parallel to X3
x1	**Always** coincides with the centroidal axis of the beam; the positive direction is from the start node (JA in the geometry tables) to the end node (JB)	As in general case
x2	As x1 and x3 are known, x2 is determined from the right-hand rule	**Always** parallel to the global X2 axis
x3	**Always** parallel to the global X3 axis	As x1 and x2 are known, x3is determined from the right-hand rule

If the angle between the local x1 and the global X3 axes is greater than 0.006°, the axes are assumed NOT parallel. As angle of this magnitude can result from computer inaccuracy, the local x3 axis direction of all beams parallel to X3 should be specifically defined.

Examples:



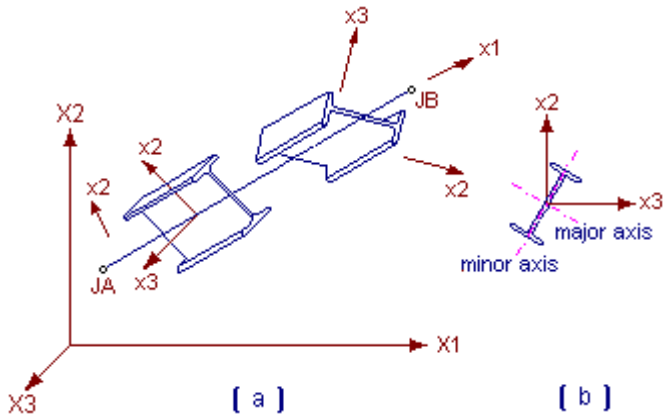
User-defined local axis directions:

As explained above, the program automatically defines default local axis directions for each beam.

STRAP always assumes that the major/minor beam axes are parallel to the local x2/x3 axes, i.e. the properties of all beams are defined relative to the local coordinate system.

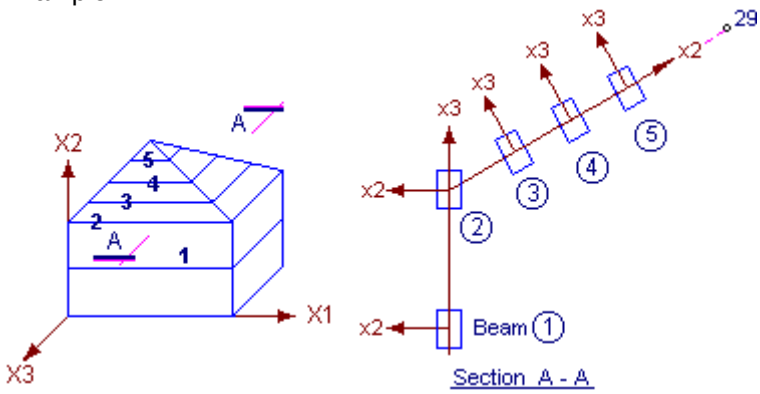
It is apparent from the example in the figure below that the beam section can be oriented in an infinite number of angles about the x1 beam axis and so the default local axes directions may not be correct.

The beam in Figure (a) below lies on the X1-X2 global plane; by default its local x3 lies parallel to the global X3 and the local x2 axis lies on the global X1-X2 plane. If the beam is a component of a sloped roof, for example, and must be aligned as shown in Figure (b), then **the local x2/x3 axes must be rotated to coincide with the major/minor axes**.



Options are available for aligning the local x2 or x3 axes with an existing node or any user-defined plane.

Example:



define the local axes of beams 1 to 5 as shown.

- beams 1 and 2: specify that the local x2 axis is parallel to the global X1-X3 plane.
- beams 3,4 and 5: specify the local x2 axes as pointing in the direction of node 29.

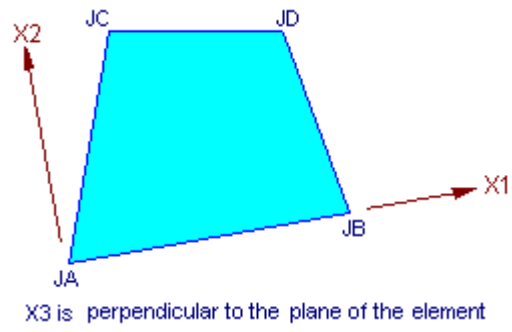
The x3 axes are determined by the program according to the right-hand rule.

1.8.2.2 Quadrilateral Finite Elements

Each two-dimensional finite element has a local coordinate system associated with it.

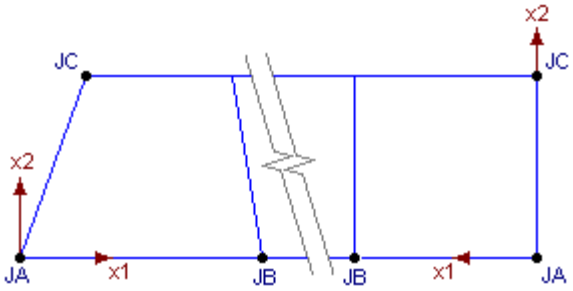
The local x1 and x2 axes **always** lie in the plane of the element and x3 is **always** perpendicular to the element.

The directions of the local axes are determined by the location of the element corner nodes.



- the local x1 axis lies along the edge of the element formed by nodes JA and JB and is positive in the direction of JB, where JA and JB are the first two corner nodes defined by the user.
- x2 is perpendicular to x1 and points in the general direction of JC, the third node defined.
- the x3 axis direction is determined by the right-hand rule.

The following figure shows a situation that can easily occur; the x1 axes of the adjacent elements point in opposite directions while the x2 axes point in the same direction; therefore the x3 axes of these elements point in opposite directions. In such a case, the sign of the results will be opposite.



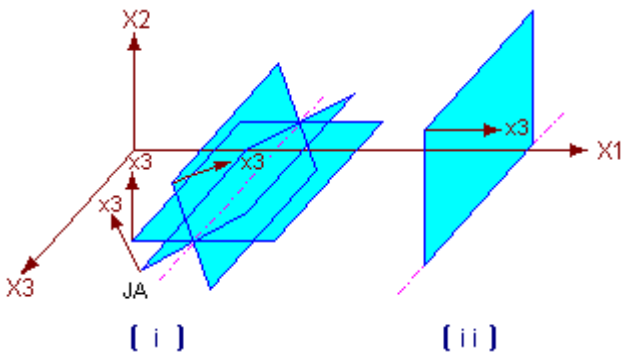
When elements are defined in the Graphic Mode, the program automatically ensures uniformity in the local x3 axis direction for adjacent elements in order to prevent confusion in the results. **The +x3 direction will always point in the general positive direction of the global +X3 axis** (except for the special cases listed below). The program reverses the x1 direction if necessary by interchanging the order of first two nodes.

To summarize the local axis selection in the Graphic Mode:

- the local x1 axis lies along the edge of the element formed by nodes JA and JB, where:
  - if a single element is defined, JA and JB are the first two nodes selected by the user.
  - if a grid/line of elements is defined, JA-JB are the nodes on the edge most parallel and closest to the base line.
- the +x2 axis lies in the element plane perpendicular to x1 and points in the direction of the other nodes.
- the **direction** of x1 is from JA to JB. The program determines the direction of the +x3 axis using the right-hand rule. If +x3 does not point in the direction closest to the global +X3 axis, the program interchanges the JA and JB nodes.

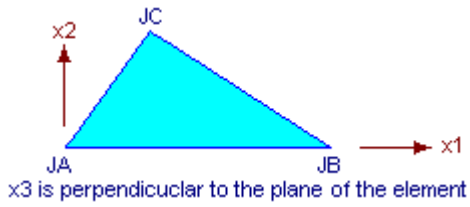
Special cases:

i. the element plane is parallel to the X3 axis: +x3 points in the direction closest to the global +X2 axis.
ii. the element lies parallel to the X2-X3 plane: +x3 points in the direction closest to the global +X1 axis.



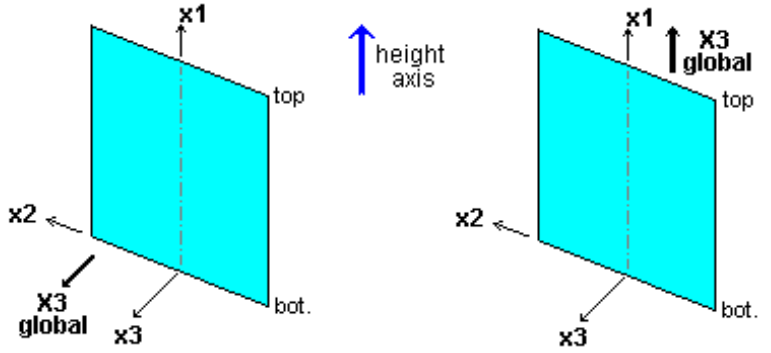
1.8.2.3 Triangular Finite Elements

The definition of local axes is similar to that for quadrilateral elements.



1.8.2.4 Wall elements

The default local coordinate system for wall element segments is identical to the default system for beams. The program assumes that the wall local x1 axis is parallel to the "height axis" specified when defining the wall and points in the positive direction of the height axis:



(a) General case

- **x1** - parallel to height axis
- **x3** - general X3 global direction
- **x2** - right-hand rule

(b) Special case: x1 parallel to X3

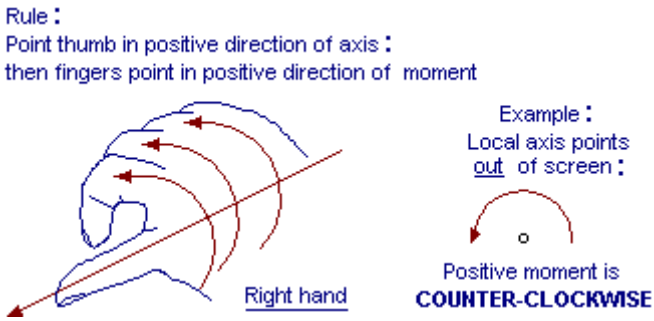
- **x1** - parallel to height axis
- **x2** - general X2 global direction
- **x3** - right-hand rule

The default local axes cannot be revised.

1.8.3 Sign Conventions - Forces and Moments

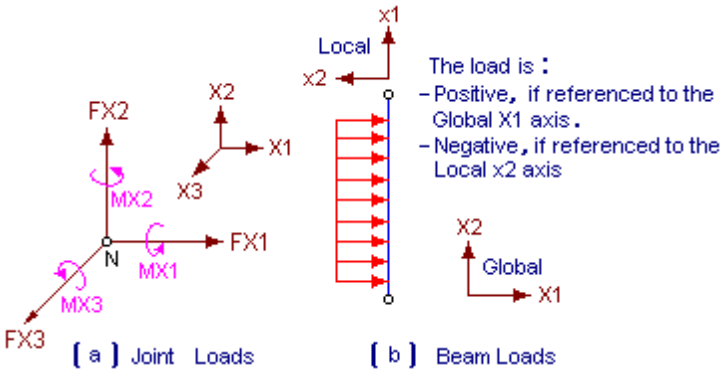
Forces and moments are referenced to a local or global coordinate axis:

- Forces: positive forces point in the positive direction of the relevant coordinate axis.
- Moments: Moments act **about** a local or global coordinate axis. The sign of the moment is determined by the following right-hand rule:

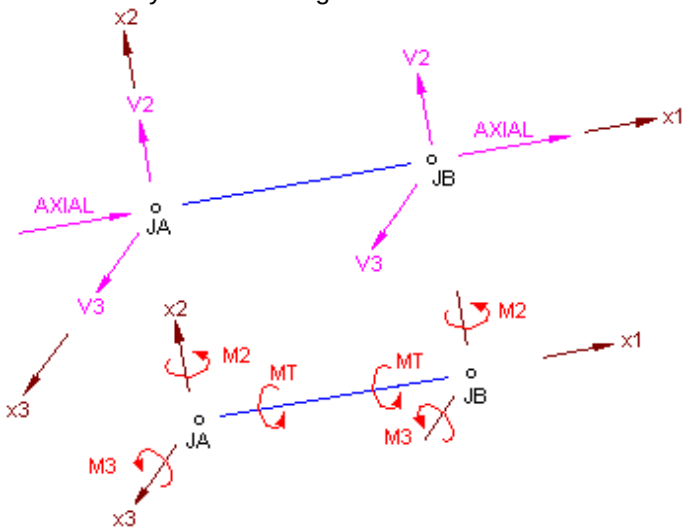


Examples:

- **Loads**
  Joint loads are **always** defined relative to the global coordinate system. Beam loads may be defined relative to either the global coordinate system or to the beam local coordinate system..

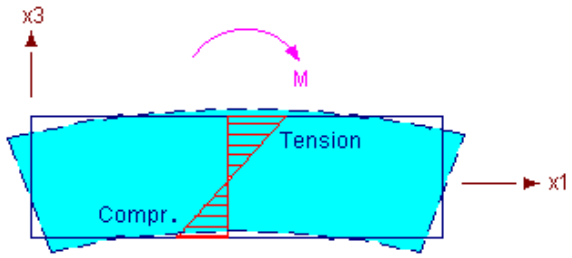


- **Beam Results (Tabular)**
  Moment, shear and axial force results for beam elements are always displayed relative to the beam local coordinate system. The sign conventions are:



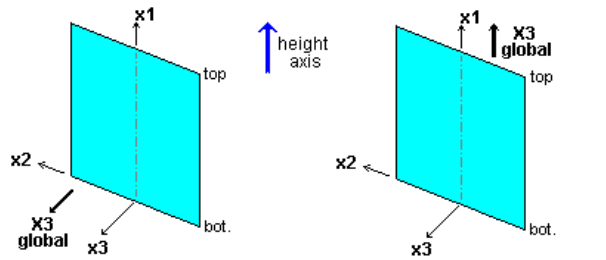
- **Finite Element Results (Tabular)**
  Moments, forces and stresses for finite elements are displayed relative to the element local coordinate system. Forces and stresses are positive if acting in the positive direction of the parallel local axis.

  Referring to the equations in 5.4.1.2 - Element results sign conventions, a positive moment creates tension on the +x3 surface of the element.



  Note that ***STRAP*** 's graphic postprocessors will often unify and reverse the signs to display the results according to accepted engineering sign conventions. Refer to 5.3 - Element result types for more details.

- **Wall elements**
  Moment, shear and axial force results for wall elements are always displayed relative to the wall local coordinate system. The sign conventions are:



(a) General case
- **x1** - parallel to height axis
- **x3** - general X3 global direction
- **x2** - right-hand rule

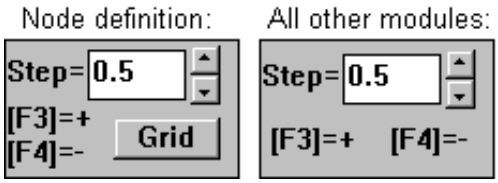
(b) Special case: x1 parallel to X3
- **x1** - parallel to height axis
- **x2** - general X2 global direction
- **x3** - right-hand rule

** Blank page **

1.9 General options

1.9.1 Step

The speed is called the **Step** and is the distance the moves on the screen each time the mouse is moved the minimum distance or an arrow key is pressed. The current **Step** value is continuously displayed in the Data Options Area.



There are three methods available for revising the Step:

- Move the into the **Step=** text box. Type in the correct value.
- Press F3 to increase the Step or F4 to decrease the Step.
- Click the buttons.

For node definition only:

Click the Grid button to display a grid of dots on the screen; the dots will be spaced at the current Step interval in both screen directions. The jumps from dot to dot when the mouse is moved or when an arrow key is pressed.

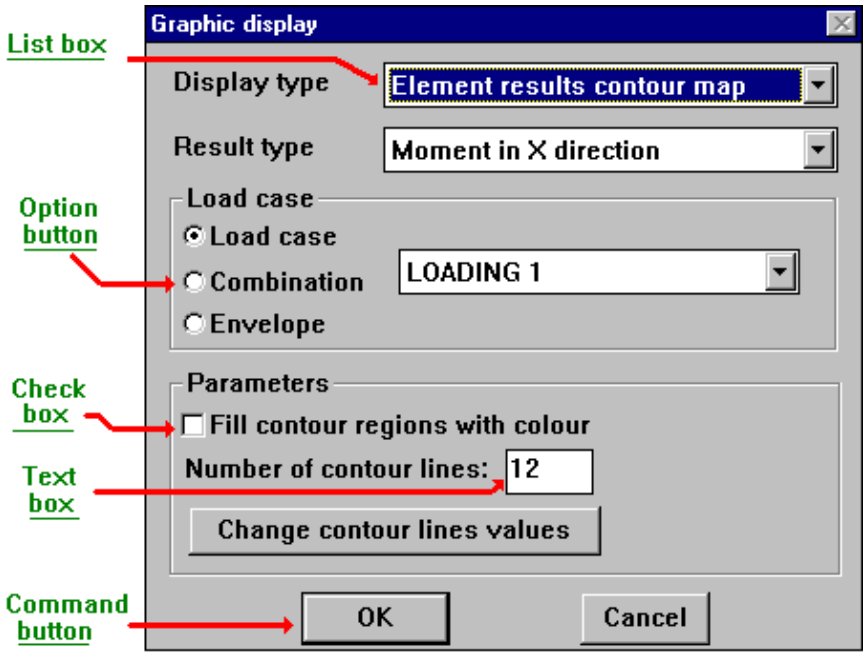
- To display the Grid automatically every time a node definition option is selected, refer to 1.4.2 - Setup - Miscellaneous.
- The grid is automatically updated every time the Step value is revised.
- If the step value is small relative to the screen dimensions, the density of the dots will be too great. In such cases the program will display the dots with a spacing equal to a multiple of the Step value; several clicks of the arrow key will be required to move from a dot to the adjacent dot.
- A cylindrical grid will be displayed when a cylindrical coordinate system is in effect (note that the mouse can be used for a cylindrical coordinate system only when the Grid is displayed).

1.9.2 Dialog boxes

STRAP uses Windows "Dialog Boxes" to request data and to provide information. Most dialog boxes contain options, each one asking for different information. After all required information is supplied, a "Command Button" is pressed to carry out the command and return the program to the previous menu or dialog box.

Dialog boxes also provide information, warnings and error messages.

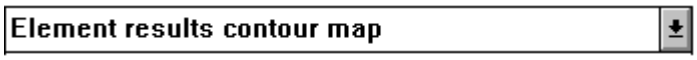
A typical "Windows" dialog box contains several types of standard option styles. For example:



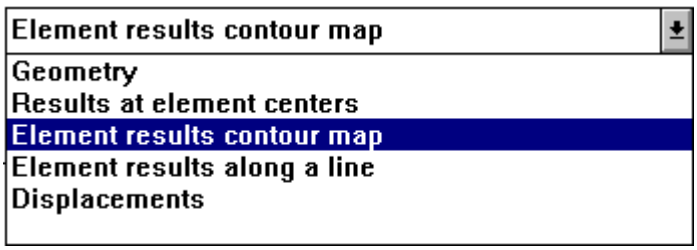
List box:

A list box displays a column of available choices. A "drop-down" list box displays only the current option along with an arrow at the end of the line.

For example, the "Result type" option in the above dialog box is displayed as:



When the arrow button is pressed, the list box opens up to:



The current option is highlighted.

Option button:

Option buttons represent a list of mutually exclusive options, i.e. only one of them can be selected. In our dialog box example, we can select to display the results for either a load case, load combination or an envelope of cases/combinations.

The selected option is highlighted by a black dot in the button . In our example, the program will display the result for a load case.

To select a new option, place the arrow on the option you want and click the mouse button.

Check box:

Check boxes represent options that can be turned on or off.

When the option in the check box is turned on, the box is displayed as ☑. In our example, the result diagrams will be hatched.

To revise the option, place the arrow on the check box and click the mouse button.

Text box:

A text box is a rectangle into which you can type information.

In the dialog box above, the number of contour lines is defined in a text box as 12.

To revise text in a text box:
- use the mouse to move the arrow to the text box and click the left mouse button
- edit the text in the box.

Command button:

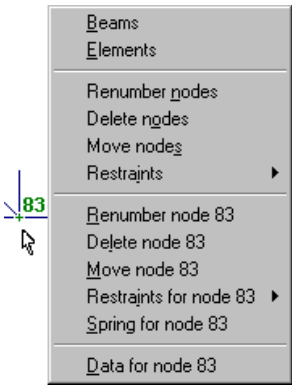
Command buttons initiate an immediate action. In the dialog box above,

- OK displays the graphic results according to all other options selected in the dialog box.
- Cancel closes the dialog box; the option is canceled and all options are returned to the values present when the dialog box was entered.

1.9.3 Shortcut Menus

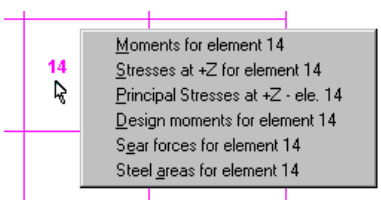
A Shortcut Menu is a useful pop-up menu that provides options specific to a certain object in the model. ***Right-click*** an object (such as a node or element) to display the Shortcut Menu for that particular object.

- Geometry:



Right click on nodes, beams or elements. For example, a right-click on a node displays a Shortcut Menu that lets you select all node related options or display data for the selected node.

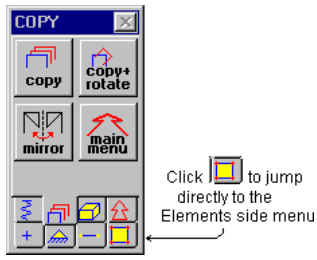
- Loads:
  a right-click on a beam displays a Shortcut Menu with beam load definition options, etc.
- Results:


14
Moments for element 14
Stresses at +Z for element 14
Principal Stresses at +Z - ele. 14
Design moments for element 14
Sear forces for element 14
Steel areas for element 14

a right-click on a node/beam/element displays a Shortcut Menu with options for displaying tabular results for the selected item.

1.9.4 Side Menus

Menu tabs are found at the bottom of all geometry 'side menus'. Clicking a tab moves you directly to the side menu of the option selected without going through the main menu. For example:



1.9.5 Help

STRAP provides the User with extensive on-Line Help; the entire User's Manual is part of the program and may be displayed at any time.

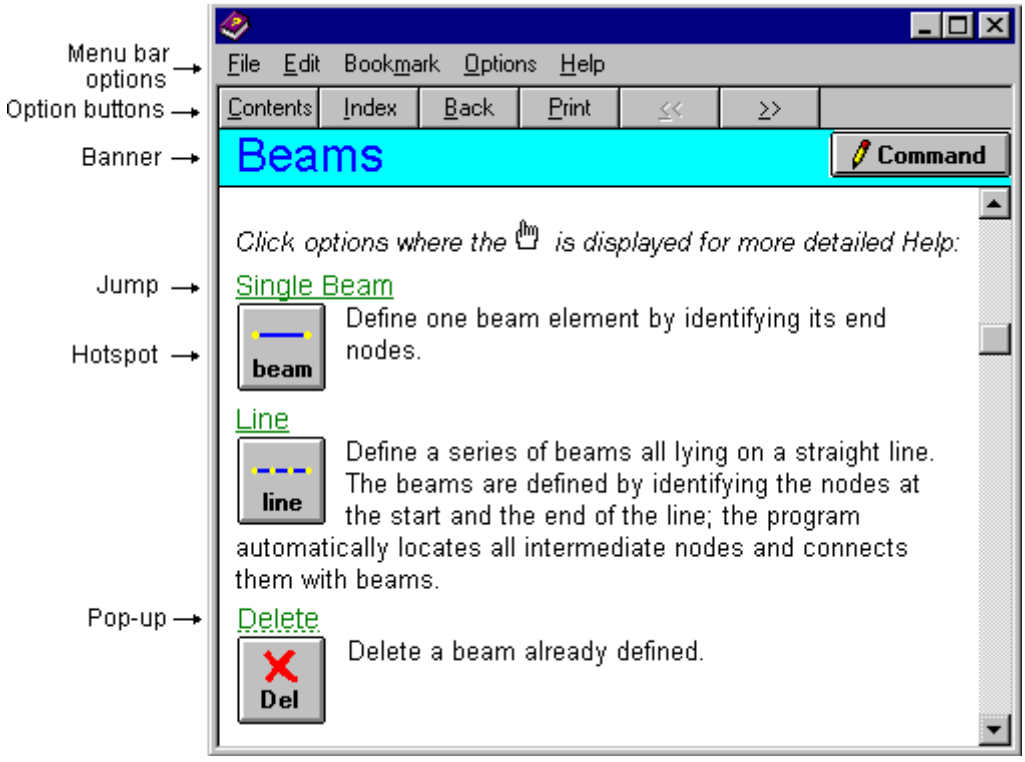
The Help for each program item is displayed in a separate topic; Help displays one topic at a time on the screen. Many options are available for locating and jumping to different topics.

There are two ways to request Help while running the program:

- Context related help - Help for the current option will be displayed on the screen.
  press the F1 key
- Display the entire User's Manual:
  Click the **Help** option in the menu bar at the top of the screen and select **Contents** in the pulldown menu; the program will display the Help for the Main menu of the current module (e.g. Geometry main menu).

  To display the Table of Contents for the entire *STRAP* User's Manual, click the Contents button.

For example, the Help topic for the Geometry beam definition menu:


Menu bar options
Option buttons
Banner
Jump
Hotspot
Pop-up
File Edit Bookmark Options Help
Contents Index Back Print << >>
Beams
Command
Click options where the is displayed for more detailed Help:
Single Beam
beam
Define one beam element by identifying its end nodes.
Line
line
Define a series of beams all lying on a straight line. The beams are defined by identifying the nodes at the start and the end of the line; the program automatically locates all intermediate nodes and connects them with beams.
Delete
Del
Delete a beam already defined.

The following Help options are available:

- Jumps, Hotspots and Pop-ups
  These hyperlink options enable you to jump directly between related topics. The cursor automatically changes to a hand when it moves over a hyperlink location. Click the mouse to display the Help for the related topic.

  Pop-up text is written over the existing help topic and is removed from the screen by clicking the mouse. Pop-up links are underlined with a dashed line.

Option buttons:

- Contents The Table of Contents of the entire STRAP User's Manual is displayed on the screen
- Index Locate information on any topic, identified by a "keyword":
- Back Return to the topic previously displayed. Clicking this button allows you to retrace your path through the Help topics you have already displayed.
- Print To print the topic currently displayed.
- << >> Display the previous/following topic in a series of related topics. The buttons are grayed out when you reach the first/last topic in a sequence.

Banner buttons:

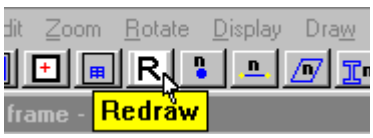
- Command for geometry and load definition options only: This button is displayed only when there is a corresponding Command Mode for the currently displayed topic. Clicking the button will display an explanation on the command format.

1.9.6 Icon bar

The icon bar provides a shortcut method for accessing ***STRAP*** options.

Each icon on the bar represents a different option. Click the icon once to run the option.

Hold the cursor on the icon for several seconds to display the option description. For example:



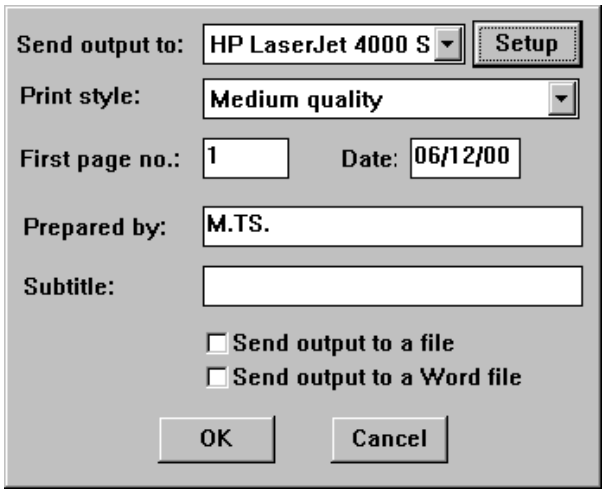
Note that you can create new icon bars or customize existing bars. Refer to 1.4.4 - Setup - icon bars.

1.9.7 Undo

Clicking the **Undo** option in the menu bar automatically cancels the last definition; the graphic display is immediately redrawn. Note that **Undo** may be pressed repeatedly, but will only cancel commands from the current session.

1.9.8 Print tables

The following menu appears when you request to print tables directly to the printer or to a file.



General options:

Refer to 1.9.9 - Print drawing.

Word format file

The tables may be saved in an MS Word RTF format file (Rich Text Format) and may be imported into any wordprocessor or program that recognizes this format.

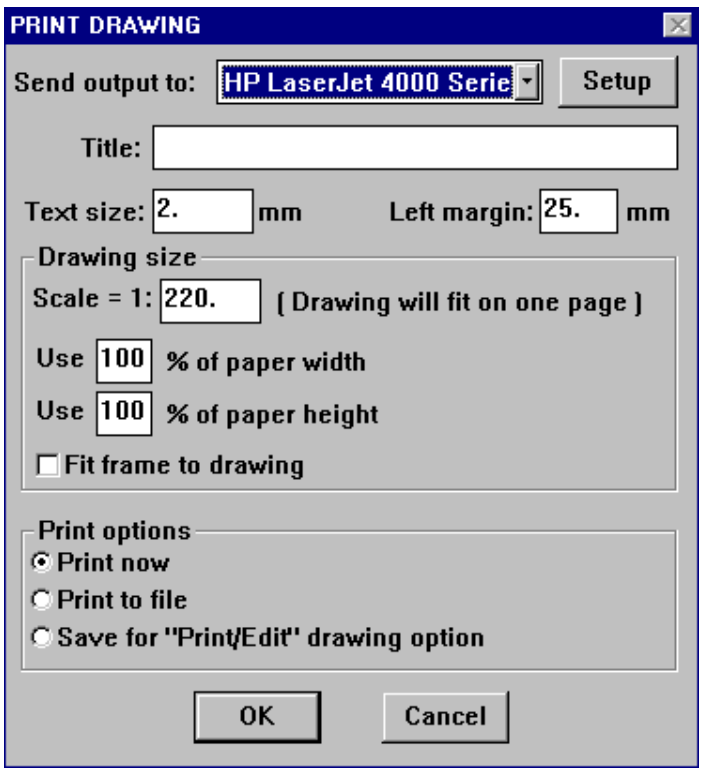
Specify a file name. The default extension is .RTF.

Note:
- a company logo may be printed at the top of every page. Refer to 1.4.3 - Setup - Print parameters.

1.9.9 Print drawing

Use this option to print the current display directly to the printer or to a file. The printed display will be enclosed in a frame and will include a header.

1.9.9.1 Regular drawing:



Setup

Specify general information for the output device selected:
- paper size
- graphic resolution
- etc.

Click the OK button to start printing.

Send output to:

Select the output unit, e.g. printer, plotter, etc.

The devices must be installed by the "Printers" option in the Windows "Control panel"

Note:
- Graphic output: to create a DXF file, select **MetaFiles** (WMF format) or **Enhanced Meta files** (EMF format) in this option, then select **Convert Metafile to DXF** in the **Files** option of the ***STRAP*** main menu.
- Tabular output: to create a 'delimited file' (data items separated by a delimiter character and not by spaces) suitable for import by most spreadsheet programs, select **Delimited file** in this option.

To specify the delimiter character, refer to 1.4.2 - Setup - Miscellaneous

Title/Text size:

- enter the name of a title that will be printed at the top of the drawing.
- specify the text size in mm. The size is used for beam numbers, node numbers, load and result values, etc.

Margin

Specify the left margin width. The program will recalculate the default scale or the number of pages required whenever a new margin value is entered.

The value is added to the default margin for the printer/page size.

Drawing - size

Specify the scale for the drawing. The program initially assumes that the drawing will fit exactly on one page (based on the paper size in the Setup option), calculates the corresponding scale and displays it as the default scale.

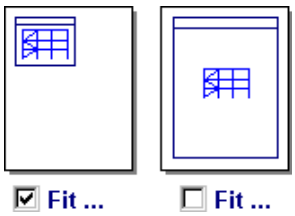
There are two methods for changing the drawing size:

- specify the scale
- specify the percentage of paper width/height to be used. Note that the program always maintains the vertical/horizontal drawing ratio, so it is sufficient to revise only one of the percentages.

If a scale larger than the default scale is specified, the drawing will be automatically printed on several pages, which can then be pasted together.

Fit frame to drawing:

Specify the location and size of the drawing and its frame:



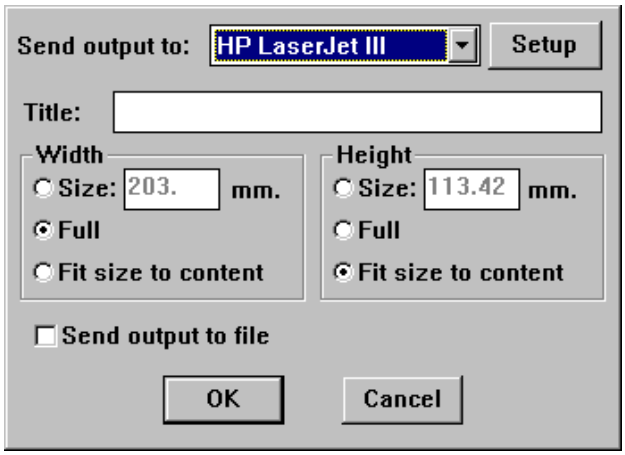
Print options

- **Print now**
  To print the current drawing immediately
- **Print to file**
  To send the drawing to a file (not required if **Metafile** is selected in **Send output to**:). The program will prompt for a file name. Note that these drawings cannot be edited using the **Print/edit saved drawing** option.
- **Save for "Print/edit drawing" option**
  To save the drawing so that it can be edited and printed using the Print/edit saved drawing option. These drawings can only be printed by *STRAP* and not by other programs or utilities.

Click the OK button to start printing.

1.9.9.2 Rendered drawing:

Use this option to print the current rendered display directly to the printer or to a file. The printed display will be enclosed in a frame and will include a header.



Width / Height

Select the height and width of the printed drawing.

Note that Windows will distort the image if you do not maintain the original proportions.

Select one of the following options for width and/or height:

- **Size** Specify the actual dimension in millimeters.
- **Full** The drawing will fit exactly into the page width height.
- **Fit size** The program will automatically calculate the dimension required to maintain the original drawing proportion.

Note: to maintain the original proportions, adjust either the width or the height while specifying **Fit size to content** for the other dimension.

All other options

Refer to 1.9.9.1.

1.9.10 Print/edit drawing

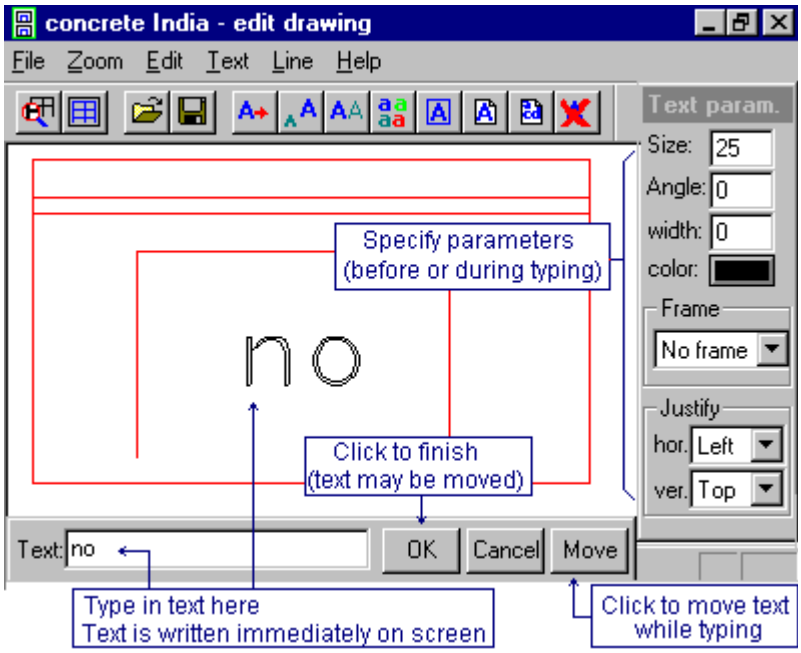
The drawing editor module contains options for modifying and adding text and lines to the graphic displays in *STRAP*. All drawings may be printed/plotted.

The main features of the program are:

- The module can differentiate between the different types of text and lines - node numbers, beam numbers, titles, etc. Hence a property (size, color, etc) can be modified at the same time for all texts in a type.
- The module recognizes the paper size of the selected printer/plotter and allows several drawings to be arranged on the page

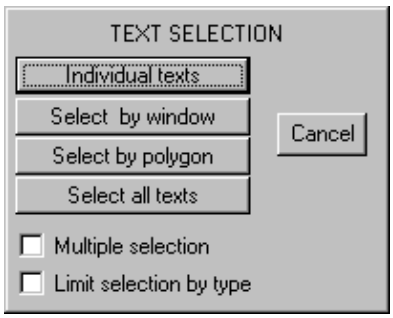
New text / Revise text

- Type and locate a line of text on the drawing according to the text parameters.
- Revise the text or properties of selected text:
  - select text using the standard text selection options
  - revise the text or any of the properties - size, angle, pen width, color, frame.



Select text

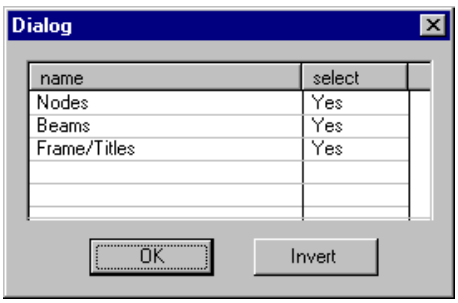
Select one or more texts using any of the following options:



- Select text - limit by type

  The program differentiates between the different text "types" on the drawing and allows the user to select texts of a certain type(s) only.

  For example, if you want to change the color of all beam numbers:



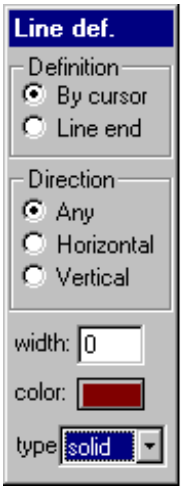
- Set **Nodes** and **Frames/titles** to **No**

- and click **OK**.

New lines

Add a new line to the drawing:

- specify the location of the line start and the line attributes:



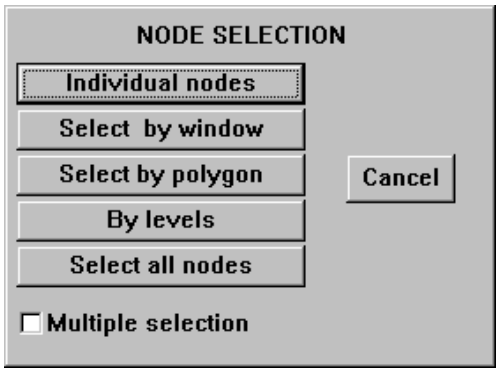
- The line may start at any location on the drawing or at the start/end of an existing line.
- Draw the line in any direction or draw only the horizontal/vertical projection of the line to the end point.
- Specify the line width (0 = 1 pixel)
- Select the line colour from the colour palette
- Specify the line type

- Move the to the location of the line start (or highlight an existing line start/end) and click the mouse.
- Specify the location of the line end. Note that the start and end of the line may be defined using different options in the menu above.
- Continue to define segments using the end of the previous line as the start of the next line, or double-click the mouse to stop.

1.9.11 Entities Selection

1.9.11.1 Node Selection

Many options include instructions to select one or more nodes.



Individual Nodes

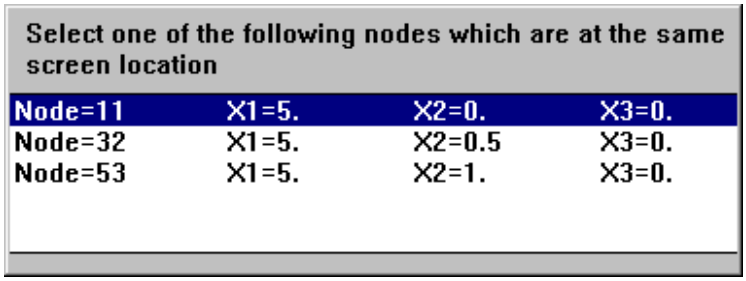
Select individual nodes by moving the alongside each node until it is highlighted by the ■ rectangular blip ; click the mouse.

The number of the highlighted node is always displayed at the left-hand side of the Dialog box:

You may also type in the number of the node to be selected, in the form of a "list".

When all the nodes have been selected, press End selection or click the mouse without moving the .

In space models, more than one node may be at the same **screen** location and so will be "hidden" from the viewer (the coordinate perpendicular to the screen of these nodes is not identical). In such a case, the program will display a list of nodes at that location and request the user to select one.


Select one of the following nodes which are at the same screen location
Node=11 X1=5. X2=0. X3=0.
Node=32 X1=5. X2=0.5 X3=0.
Node=53 X1=5. X2=1. X3=0.

Move the cursor to the line with the correct node and click the mouse.

Select by window

Define a rectangular window by pointing to its lower-left and upper-right corners with the . The program will automatically identify all nodes located in the window.

In space models, more than one node may be at the same **screen** location and so will be "hidden" from the viewer. In such a case, the program will select **all** of the nodes at that location.

Select by polygon

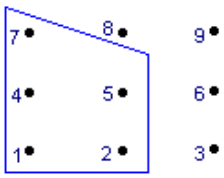
Define a polygon by pointing to its corners with the . The program will automatically identify all nodes located in the polygon.

The polygon is constructed as a 'rubber-band' stretched around the defined corners:

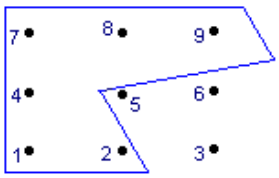
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the .
- In space models, if more than one node is at the same **screen** location, the program will select **all** of the nodes at that location.

Example:

select nodes 1,2,4,5 and 7

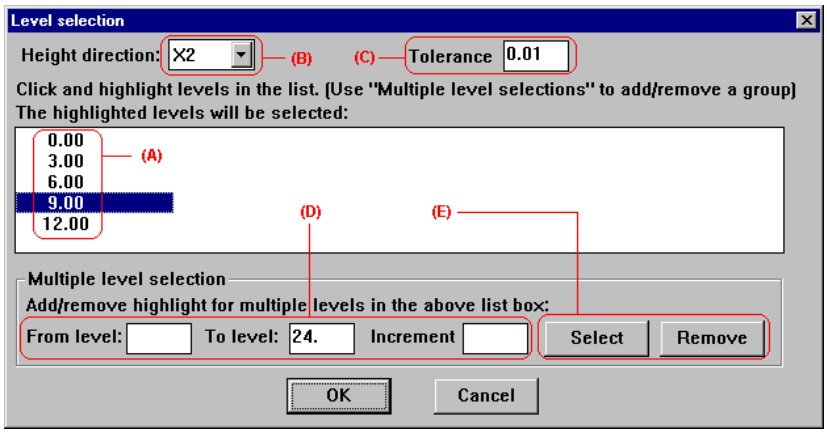


1,2,4,7,8 and 9



By levels

Select all nodes at selected levels (coordinates with nodes).



- The program displays a list of the levels parallel to one of the global axes (A). A different global axis may be selected in the Height direction list box (B).
- Select levels by clicking and highlighting them (A)
- Equally spaced levels may be selected by entering the coordinate of the start level, end level and increment in edit boxes (D). Click Select (E) to highlight these levels in the list or Remove (E) to remove the highlight.
- Click OK; the program will identify all nodes at the selected levels

Note:
All nodes within the ± tolerance distance (C) will be selected.

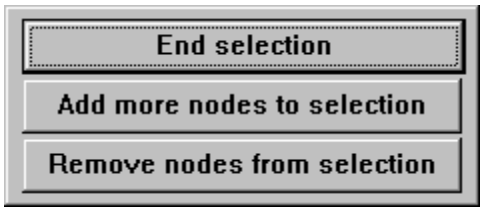
Select all nodes

All nodes in the model will be selected.

Note:
- nodes not displayed because of the **Zoom** option will be selected.
- nodes not displayed because of the **Remove elements** option (all beams/elements attached to these nodes are removed) will not be selected.

Multiple selection

Turn on the ☑ checkbox if you want to define several windows, polygons or lists for the same command. After every window, etc., the following menu is displayed:



End selection - Continue without selecting more nodes

Add more nodes - Define another window, polygon, etc. for the same command.

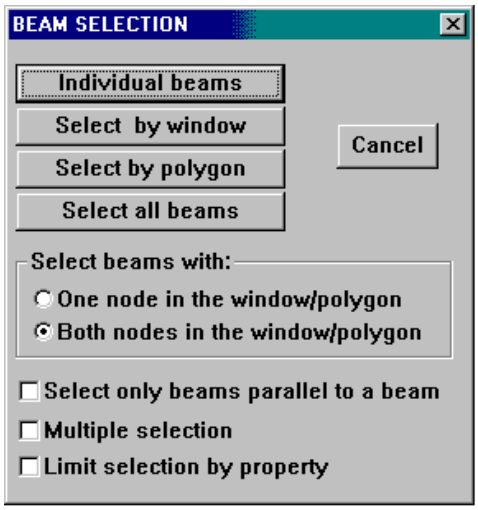
Remove nodes -- Delete members from the list, window, etc. already defined for this command.

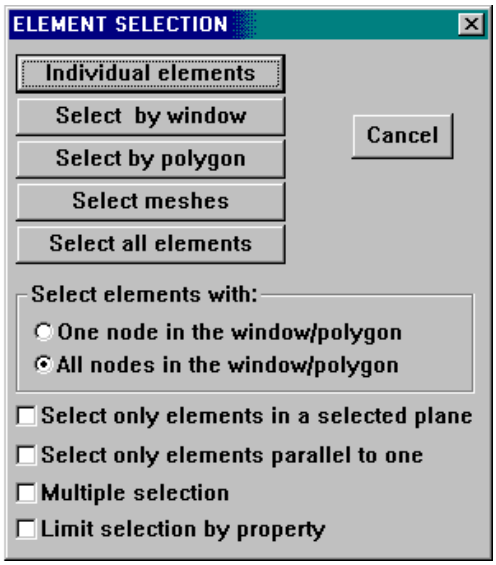
1.9.11.2 Beam/Element Selection

Many options include instructions to select one or more elements.

When selecting an element, the element nearest to the is highlighted with a rectangular blip ■. The number of the highlighted element is always displayed at the left-hand side of the Dialog box.

You may also type in the number of the element to be selected.





Individual elements

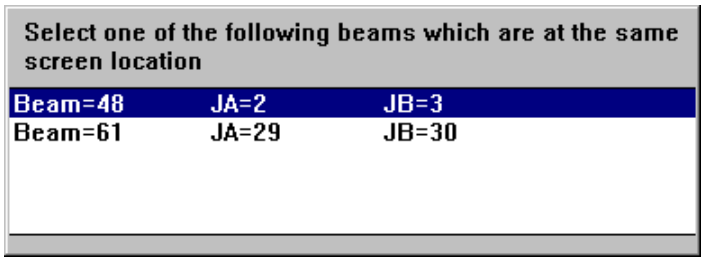
Select a single element only by moving the alongside the element until it is highlighted by a rectangular blip ■; click the mouse.

The number of the highlighted beam is always displayed at the left-hand side of the Dialog box:

You may also type in the number of the beam to be selected, in the form of a "list".

When all the beams have been selected, press End selection or click the mouse without moving the .

In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program will display a list of elements at that location and request the user to select one:



Highlight the line with the correct beam and click the mouse.

Select by window

Define a rectangular window by pointing to its lower-left and upper-right corners with the . The program will automatically identify all elements with either one or all nodes located in the window (refer to “Select beams with....” below)

In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such cases the program will select **all** elements at that location.

Select by polygon

Define a polygon by pointing to its corners with the . The program will automatically identify all elements located in the polygon. The program will automatically identify all elements with either one or all nodes located in the polygon (refer to **Select beams with**)

The polygon is constructed as a 'rubber-band' stretched around the defined corners:
- At least three corners must be selected.
- the program automatically connects the last corner defined to the first corner defined.
- press [Esc] (right mouse button) to delete the previous corner.
- to end the polygon definition, click the mouse without moving the

In space models, more than one element may be at the same screen location (only the coordinate perpendicular to the screen is not identical). In such a case, the program will select **all** of the elements at that location.

Select all beams/elements

All beams/elements in the model will be selected.

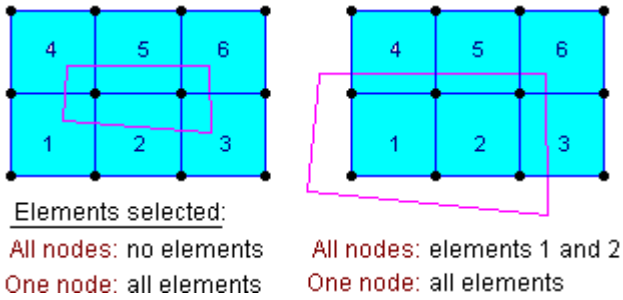
Note:
- beams/elements not displayed because of the **Zoom** option will be selected.
- beams/elements not displayed because of the **Remove beams/elements** option will not be selected.

Select beams/elements with:

One node in the window/polygon:
all elements with at least one end/corner node in the window/polygon will be selected.

Both nodes in the window/polygon:
only elements with **all** end/corner nodes in the window/polygon will be selected.

Examples:

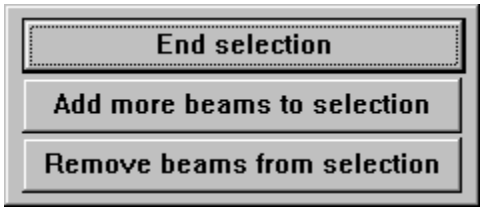


Select only beams/elements parallel to a beam/elements

You may impose a further limit that the beams/elements selected will be only those parallel to a specified one. This option allows you, for example, to define a window around an entire frame but to select only the beams or columns.

Multiple selection

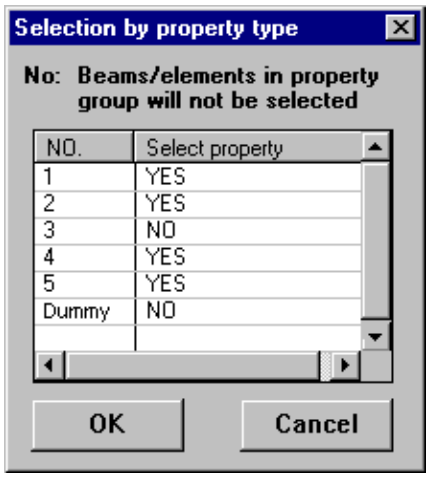
Turn on the ☑ checkbox if you want to define several windows, polygons or lists for the same command. After every window, etc., the following menu is displayed:



End selection - Continue without selecting more elements.
Add more beams - Define another window, polygon, etc. for the same command.
Delete beams - Delete members from the list, window, etc. already defined for this command

Limit by properties

Further limit the beam/element selection according to property groups:



YES: Beams/elements in the property group will be selected
NO: Beams/elements in the property group will **NOT** be selected

Note:
- Dummy beams/elements may be selected

Select meshes

Select **all** elements in a mesh created by the geometry **Element - Mesh** option. Move the alongside any element in the mesh until it is highlighted by a rectangular blip ■; click the mouse.

Note that all contours of all defined meshes will be highlighted with a thick line during the selection.

Elements on a plane

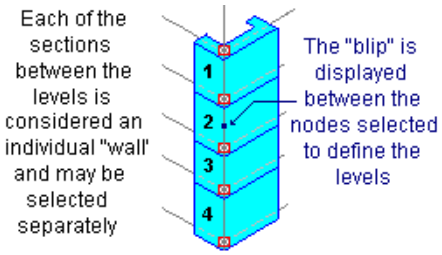
For finite elements only, you may impose a further limit that the elements selected will be only those lying on a specified plane. This option allows you, for example, to define a window around an entire model while selecting only a certain level.

The plane is identified by pointing to an existing element; only those elements lying on the same plane as this element will be selected.

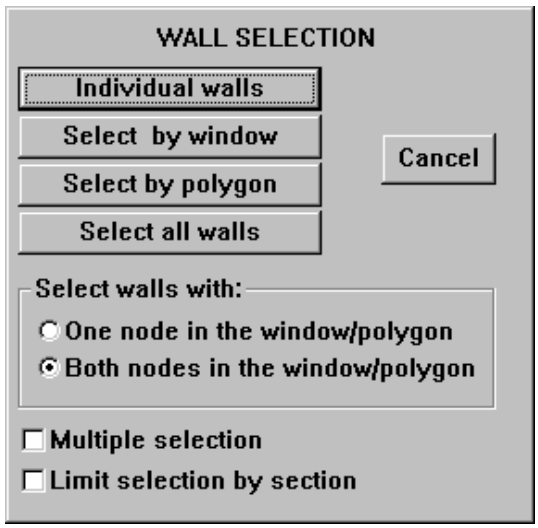
1.9.11.3 Wall selection

Many options include instructions to select one or more walls.

Wall selection is similar to beam selection. Each section between the "attachment" nodes specified when adding the wall to the model is considered a separate wall For example:


Each of the sections between the levels is considered an individual "wall" and may be selected separately
1
2
3
4
The "blip" is displayed between the nodes selected to define the levels

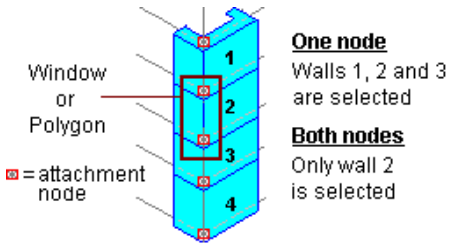
The wall nearest to the is highlighted with a rectangular blip ■. The number of the highlighted wall is always displayed at the left-hand side of the Dialog box. You may also type in the number of the wall to be selected.



Individual walls
Select by window
Select by polygon
Select all walls
Refer to beam/element selection

Select with one node/both nodes

- **One node in the window/polygon**:
  all walls with at least one end node in the window/polygon will be selected.
- **Both nodes in the window/polygon**:
  only walls with **both** end nodes in the window/polygon will be selected.

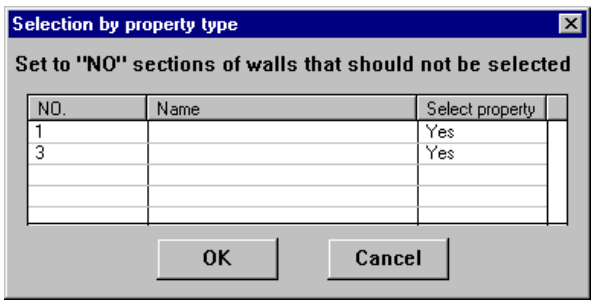


Note that the program only considers the nodes where the wall "reference point" was attached to the model:

Multiple selection
Refer to beam/element selection

Limit selection by section

Further limit the wall selection according to section type:
Yes: walls in the section group will be selected
No: walls in the wall group will **NOT** be selected



1.10 Command mode

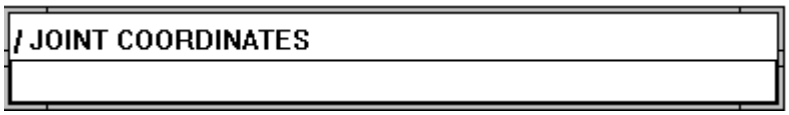
The following section explains in general how to define the model using the Command Mode. For a complete explanation, refer to the Command Mode Manual.

The commands are typed in by the user in the Command Mode Box at the bottom of the screen; the program automatically updates the graphic display of the model.

It is important to note the following:
- the Graphic Mode and the Command Mode may be used concurrently.
- every time a part of the model is defined in the Graphic Mode, the program automatically writes the equivalent command in the Graphic Mode Box. Therefore, the Command Mode Box contains a complete record of all that was defined in the current session.

1.10.1 Enter a Command

When you enter a definition type, the program automatically writes a header in the Command Mode Box. For example, when you select **Nodes**, the Command Mode Box appears as:



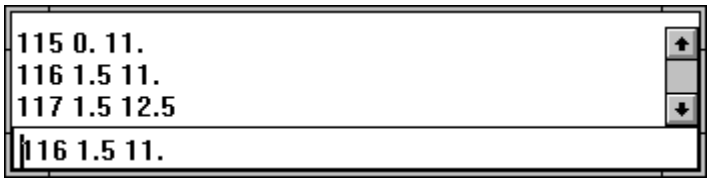
To type in a command, move the below the last line in the box so that the **I** cursor appears; click the mouse.

Type in the command in the correct format and press [Enter]; the Graphic Display will be updated.

1.10.2 Revise a Command

All commands are entered in the command box as the model is defined. The box displays only two rows and so the commands are scrolled up and disappear (the box can be enlarged as any "Windows" box). To recall scrolled commands, click the up/down arrow buttons at the right side of the box.

Example: revise the definition for beam 116.



- move the to the command and click the mouse; the command is rewritten at the bottom of the Command Box and the **I** cursor is displayed at the beginning of the command.
- Correct the command and press [Enter]; the graphic display will be updated.

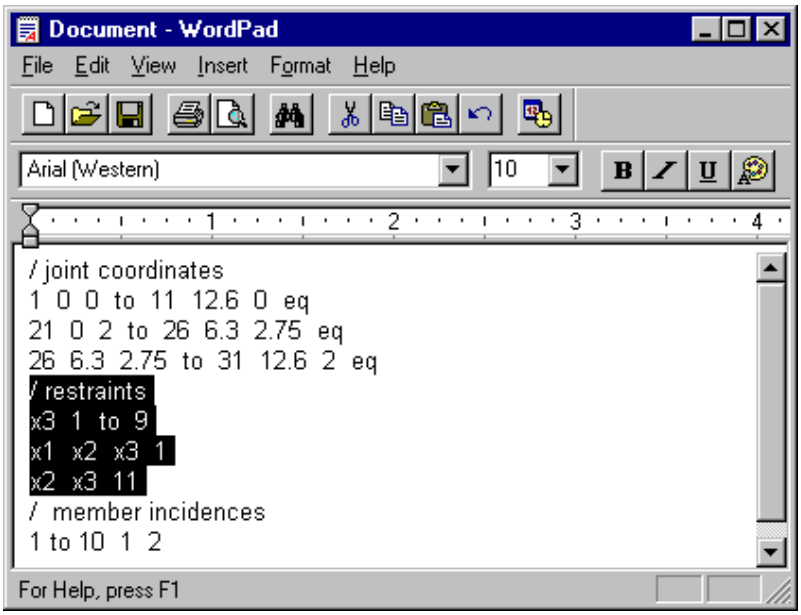
1.10.3 Retrieve a command (from the Clipboard)

Commands located in any ASCII file may be retrieved via the Windows "Clipboard".

This option is similar to the Batch Mode, but allows selected commands to be retrieved rather than the entire file.

To write the commands to the clipboard:
- run the program "Wordpad"/"Notepad"/Write", a "Windows" line editor.
- either type in the commands or retrieve an existing file using the **File** option.
- move the to the start of the block of commands; click the mouse.
- without releasing the mouse key drag the to the end of the last command in the block; release the mouse

key (the block should be highlighted).


Document - WordPad
File Edit View Insert Format Help
Arial (Western)
10
/ joint coordinates
1 0 0 to 11 12.6 0 eq
21 0 2 to 26 6.3 2.75 eq
26 6.3 2.75 to 31 12.6 2 eq
/ restraints
x3 1 to 9
x1 x2 x3 1
x2 x3 11
/ member incidences
1 to 10 1 2
For Help, press F1

- Select the **Edit** option from the top menu bar.
- Select the **Copy** option from the pull-down menu.
- The block of text is now in the clipboard; exit the "Write" program.

To retrieve the commands from the Clipboard into ***STRAP*** :
- Select the **Edit** option from the ***STRAP*** top menu bar.
- Select **Paste commands** from the pull-down menu.
- The program reads the commands, adds them to the Command Box and updates the graphic display.

If the program encounters commands with format errors or commands that generate warnings (i.e. redefinition of existing nodes), it pauses and displays the following menu:

Edit command:

The command is displayed in the Command Box at the bottom of the screen; edit the command as explained in "Revise a command". The program will then continue to the next command in the clipboard.

Ignore command:

The program will ignore the current command and continue to the next command in the clipboard.

Quit copy:

The program will ignore the current command and all following commands in the clipboard.

1.10.4 Command Format - General

All data is input in free format. There must be at least one blank space between one data value and another (including words, numerical values, and special symbols - without exception).

The commands must be in the format specified by this manual. For example, to define the self-weight of beams as a load on the structure, the manual specifies the command:

Self |X1| (f) BEAM list
     |X2|
     |X3|

The words in capital letters are keywords which must be entered exactly as they appear in the format statement.

The program normally requires only the first one or two letters of a keyword in order to identify it. Underlined letters indicate the letters that the program reads. Simplify the input by typing these letters only. For example:

S instead of SELF
B instead of BEAM

Lower-case letters indicate numerical data. In general, parameters beginning with i,j,k,l,m,n indicate integer values, and all other letters indicate decimal values.

examples:
n, n1, ... are symbols for node numbers (integers)
p, p1, ... are symbols for section dimensions (decimals)

Decimal values may also be entered exponentially. For example:

- **510** may be entered as **5.1E2** or **5.1E+2**
- **0.0037** may be entered as **.37E-2** or **3.7E-3**
  Do not leave any blank spaces between the numbers and the letter **E**.

Parameters in brackets ( ) indicate optional input. In the self-weight example above, f may be omitted.

Keywords in brackets | | indicate a choice of one of the keywords listed. In the above example, type one of X1,X2,X3 to specify the direction the loads act.

"list" indicates a list of nodes or elements in the List Format. For example:

1 9 17 20
1 3 TO 6 12 15 18 TO 30
3 TO 11 BY 2 20 TO 24 34

The last example is equivalent to entering:

3 5 7 9 11 20 21 22 23 24 34

A list can consist of up to 50 items, where " **1 TO 6** " is one item.

Typical input for the self-weight example are:

SELF X1 B 10 11 12
S X3 -1.4 B 1 TO 90

The program checks the validity of every character entered; an illegal entry will cause the computer to beep and the character will not be displayed on the screen.

** Blank page **

1.11 Batch mode

"BATCH" processing is a method where the model files are created external to the program.

STRAP has facilities for processing geometry, loads and load combinations, solving the model and generating input/output data files without using the interactive graphic interface.

- geometry, loads and/or combinations may be defined by the user by typing commands in an ASCII file using any editor program
- models created in batch mode can be added to the model list using the **Add a file to model list** option in the **File** pulldown menu, and can then be solved, etc.
- alternatively, the model can be solved using the *STBatch* utility (refer to the *STBatch* manual)
- for all models, input/output ASCII data files in user specified format can be generated by the *STBatch* utility

1.11.1 Batch mode - Geometry

There are two methods for defining ***STRAP*** geometry via external files:

- Using the Clipboard (refer to 1.10.3)
- Creating and importing an ASCII file

The file format is as follows:

1st line:

```
| REPLACE    |
| ADD        |
```

where:

REPLACE = the program will use this file instead of the existing geometry file.
ADD = the program adds the commands in this file to the data in the existing binary geometry file (GEOMnnn.DAT).

(Do not write IGNORE in this line)

Command Lines:

All commands are in the regular format. Before the first line of each command type, a header must be entered on a separate line. The headers are:

```
/ JOINT COORDINATES
/ RESTRAINTS
/ PROPERTY NUMBER
/ BEAM END RELEASES
/ MEMBER INCIDENCES
/ MATERIAL
/ PROPERTY DEFINITION
/ SPRINGS
/ DUPLICATE A BLOCK
/ UNITS force length
```

Notes:

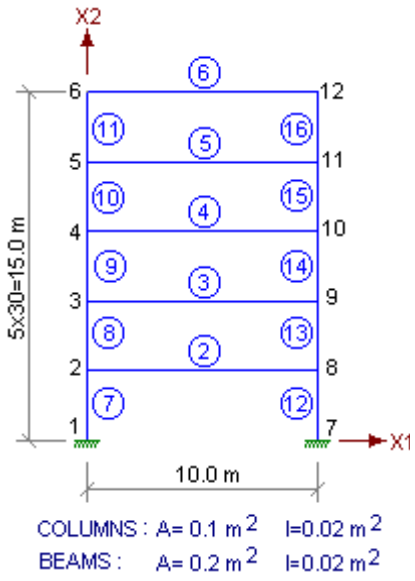
- There must be a space after the "/".
- Lines beginning with " ; " are comment lines and are ignored by the program.
- No blank lines are allowed.
- The order is not important; the commands of any type may appear in scattered groups as long as each group begins with the header.

If REPLACE is used, the program does not read the binary geometry file and hence does not know what the model is 'plane' or 'space'. It assumes that the model is 'space' and expects three coordinates in every node definition command. Type the command COORD 2 on a separate line (after / JOINT COORDINATES) if the following commands contain only two coordinates.

If the program discovers a format error in a command when reading the geometry file, it exits and displays a message. The error/warning messages are written to a file "ERR1.LST" that may be displayed or printed.

Refer also to GEOINnnn.DAT. (after the following example)

For the following example, the ASCII file would be:




```
REPLACE
/ JOINT COORDINATES
COORD 2
1 0 0 TO 6 0 15 EQ
7 10 0 TO 12 10 15 EQ
/ RESTRAINTS
X1 X2 X6 1 7
/ PROPERTY NUMBERS
1 1 TO 10
2 11 TO 15
/ MEMBER INCIDENCES
1 TO 5 1 2
6 TO 10 7 8
11 TO 15 7 8
/ MATERIALS
CONC
/ PROPERTY DEFINITION
1 A 0.1 I 0.002
2 A 0.2 I 0.02
```

GEOINnnn.DAT

The current geometry for each model is stored in a binary file named GEOMnnn.DAT, where "nnn" can be verified by selecting the **Display all model files** option in the **Files** pull-down menu on the main menu bar.

However, ***STRAP*** simultaneously creates for each model an ASCII data file that contains all of the geometry data that was defined interactively in the form of commands. The command formats are explained in detail in the Command Mode manual.

The file name is: GEOINnnn.DAT

This file may be edited or updated external to the program using a screen editor; ***STRAP*** can then be instructed to use this file instead of the existing geometry file (GEOMnnn.DAT) as the source for the model geometry.

When you choose the **Geometry** option in the Main Menu bar, the program reads the first line of the GEOINnnn.DAT file. If the first line is:

IGNORE : the program will ignore this file and use the GEOMnnn.DAT file only. This is the normal case because when the model is entered interactively, the program automatically writes IGNORE in the first line of the GEOINnnn.DAT file

REPLACE : the program will use this file instead of the existing geometry file, i.e it ignores the GEOMnnn.DAT file

ADD : the program adds the commands in this file to existing geometry in the GEOMnnn.DAT file. If data is defined in both files the GEOINnnn.DAT data will overwrite the GEOMnnn.DAT data.

If a format error is discovered in a command, the program exits with a warning. All warning and error messages are entered in a file "ERR1.LST" which may be displayed or printed.

After reading all of the command lines in the file, the program changes the first line back to IGNORE.

1.11.2 Batch mode - Loads

There are two methods for defining *STRAP* loads via external files:
- Using the Clipboard
- Creating a STATnnn.DAT file

An ASCII file containing all of the load commands for a model may be created and updated external to ***STRAP***.

The file name must be **STATnnn.DAT**, where "nnn" can be verified by selecting the **Display all model files** option in the **Files** pull-down menu on the main menu bar.

The file format is as follows:

1st line: ASCII

for each load case - 1st line: *load case title*

All loads must be in the command format as explained in detail in the Command Mode manual. Before the first line of each load type, a header must be entered on a separate line. The headers are:

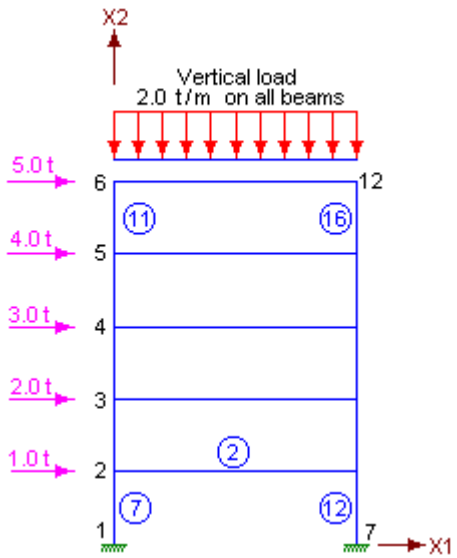
```
/ BEAM LOADS
/ JOINT LOADS
/ DISPLACEMENTS
/ ELEMENT PRESSURE
/ LOAD COMBINATIONS
/ GLOBAL LOADS
```

end of load case: / END

end of file: / END STATIC

Notes:
- In all command lines, there must be a space after the "/".
- No blank lines are allowed.
- The order is not important; the commands of any type may appear in scattered groups as long as each group begins with the header.
- The last loading case should not have a "/ END" command prior to the "/ END STATIC" command.

Example: For the following space frame example, the STATnnn.DAT file is:




```
ASCII
SELF-WEIGHT AND ROOF LOADS
/ BEAM LOADS
SELF  X3  B  2  TO  16
U  GLOB  FX3  -2.0  B  2  TO  6
/ END
WIND LOADS
/ JOINT LOADS
FX1  1.0  N  2
FX1  2.0  N  3
FX1  3.0  N  4
FX1  4.0  N  5
FX1  5.0  N  6
/ END  STATIC
```

1.11.3 Batch mode - Combinations

A file containing load combination commands can be imported into ***STRAP*** by "cutting and pasting" the commands in the Results "Define/revise combinations" option:

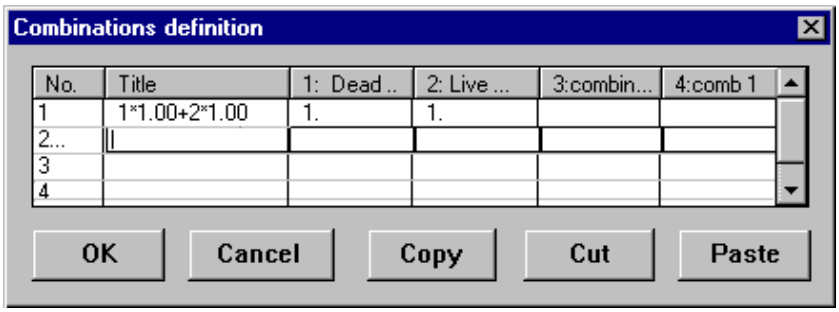
- type the combination commands in a Windows editor program such as "Wordpad" in the format:

  TITLE tit (optional)
  lc1 f1 lc2 f2 .....lcn fn..G1 fg1 ..... Gn fgn

  where:
  tit = combination title string. The program will create a default title if this line is omitted
  lcn = load case number
  fn = factor for load case 'n'
  fgn = factor for group 'n'

  Example:
  for a combination 1.4*load case 1 + 1.6 * load case 3 + 1.2 * group 2, titled "Dead + Live + Group 2", type:
  TITLE Dead + Live & Group 2
  1 1.4 3 1.6 G2 1.2

- Highlight the commands (click and hold the mouse, drag the cursor), then select "**Edit**", "**Copy**" in the menu bar
- Press [Alt][Tab] to return to ***STRAP***



- place the anywhere on the line where the command is to be written and click the mouse (if you select a line with an existing combination, then the command will be inserted above the line).
- click the Paste button

Note that multiple commands may be "cut and pasted" at the same time.

2 Geometry - Main Menu

The program displays the geometry definition screen (including the current model if an existing model was selected). The geometry definition options are displayed at the right-hand side of the screen:

To define node coordinates. Refer to 2.1.

To define restrained supports and rigid links. Refer to 2.2.

To define beam elements, including: location of the beam (between two nodes), properties, materials, pinned connections, rigid offsets, local axes directions. Refer to 2.3.

To define quadrilateral and triangular finite elements, including location of the element, properties, materials (including orthotropic). Refer to 2.4.

To define elastic supports. Refer to 2.5.

To duplicate a portion of the model at another location, including nodes, elements and properties. The copied portion of the model may be rotated or a mirror image may be created. Refer to 2.6

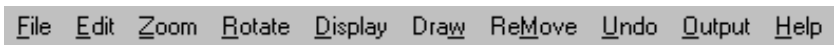
To define general solid elements, including material location and shape of the elements. Note that the elements may have 4, 5, 6, 7 or 8 corner nodes. Refer to 2.7.

To define wall elements sections and location. Refer to 2.8.

To create part of the current model in a temporary working area called a "sub-model". The sub-model may then be inserted in the main model. Refer to 2.9.

Define construction ‘stages’. Different properties and supports may be defined for each stage and beams/elements may be removed. Refer to 2.9.

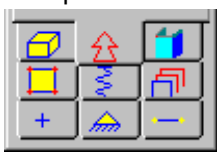
Menu bar options:



- Undo - Refer to 1.9.7
- All other options - Refer to the extended manual.

Note:

- the options can also be accessed by clicking on the small tabs below the main icons:



- refer also to the Command Mode manual for details on defining geometry by typing in commands.

2.1 Nodes

Define the location of each node in space by specifying its coordinates in the global coordinate system.

When **Nodes** are selected in the geometry Main Menu, the program displays the following options:

Define one node only. Refer to 2.1.1.

Define a series of nodes all lying on a straight line, with equal spacing between them. When defining the nodes using a cylindrical coordinate system, the nodes are equally spaced along an arc. Refer to 2.1.2.

Define a series of nodes all lying on a straight line, with varying spacing between them. When defining the nodes using a cylindrical coordinate system, the nodes are located along an arc. Refer to 2.1.3.

Define a parallelogram grid of nodes. Define a 'base' line and a 'height' line of nodes (similar to the previous two options) by specifying the location of three corner nodes. When defining nodes using a cylindrical coordinate system, this command creates a series of parallel arcs or concentric arcs.

Define a series of nodes using an equation, e.g. parabola, sphere, etc. The equation may be one supplied with the program or may be user defined. The option may also generate beams and/or elements.

Move an existing node to a new location.

Delete an existing node.

Assign a new node number to an existing node.

Select a coordinate system for node definition:
- a Cartesian plane, either one of the global planes or any arbitrary plane in space.
- a Cylindrical coordinate system.

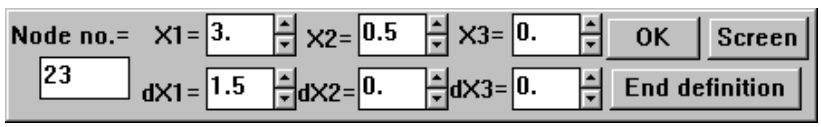
Refer to 2.1.4.

Search for locations where more than one node has been defined and delete all nodes except one, thereby 'unifying' the model at those locations.

2.1.1 Single Node

Create a single node at any location in the model.

The standard node definition dialog box is displayed:



X1,X2,X3 = global coordinates of next node to be defined
dX1,dX2,dX3 = distance from previous defined node
Node no. = number of next node to be defined

The program numbers the nodes automatically by assigning the next available number to the node being defined. The user can override the program and specify the number of the next node by moving the into the **Node no**. text box and typing in a new number.

Click:

- Screen to return the crosshair to the graphic display without leaving the current option and without creating the node/beam/etc.
- End definition to abort the current option.
- OK to create a node at the coordinates currently displayed.

Plane model:

There are two methods available:

- move the mouse so that the is located at the correct X1, X2 coordinates as displayed in the Dialog Box (adjust the Step if necessary); click the mouse.
- move the into the X1 text box in the Dialog Box. Either type in the correct value or click the buttons until the correct value is displayed.

Repeat for X2 and press OK to complete the definition.

Note that if you click and hold the left mouse button until the is in the Dialog box, then the initial coordinates in the box will be at the location when the mouse was clicked.

A node numbered with the **Node no.** is created and displayed.

Space model:

The X3 coordinate must be defined in the Dialog box:

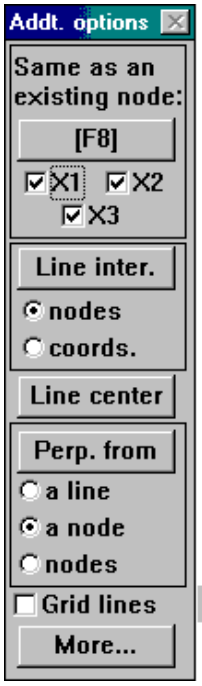
- move the into the X3 text box in the Dialog Box. Either type in the correct value or click the button until the correct value is displayed.

  Note that if you click and hold the left mouse button until the is in the Dialog box, then the X1 and X2 coordinates in the box will be those at the location when the mouse was clicked.

- Define the X1, X2 coordinates as explained in the **Plane Model** example above.

Note that the mouse can move along any plane, referred to as the Working plane. By default the program selects the Global X1-X2 plane as the working plane. However the user may select any of the other global planes as the working plane or define any plane in space by specifying three nodes.

Additional options:



- Define with same coordinates as existing node:

  Specify that one or more of the global coordinates is identical to the same coordinates of an existing node:

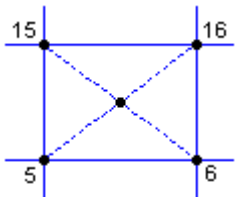
  - select the identical global coordinates by turning on their ☑ checkboxes
  - click the [F8] button
  - select an existing node:

  Move the ↖ to an existing node (the ↖ will move only in the allowable directions) and click the mouse or type in the node number in the dialog box and press [Enter].

- Define a node at intersection of lines:

  ◉ **Intersection by nodes**

  Using the ↖, define two lines by pointing to their start and end nodes; the new node are created at the point of intersection of the two lines.



For example, create the node at the centre of the bay:
first line: 5-16 second line: 6-15

◉ **Intersection by coordinates**

Using the ↖, define two lines by pointing to their start and end coordinates; the new node will be created at the point of intersection of the two lines.

- Define a node at the center of a line

  Create a new node at the centre point of the imaginary line connecting two existing nodes.
  - select the start and end nodes of the line

- Define a node perpendicular to a node/line

  A node (single node, start/end node of a line, etc) may be defined at:
  - the intersection of a line (defined by two nodes) and the perpendicular to the line from a third node
  - offset from a node and perpendicular to the line starting at that node; the node is at a specified distance or the intersection of the perpendicular with another line or arc.

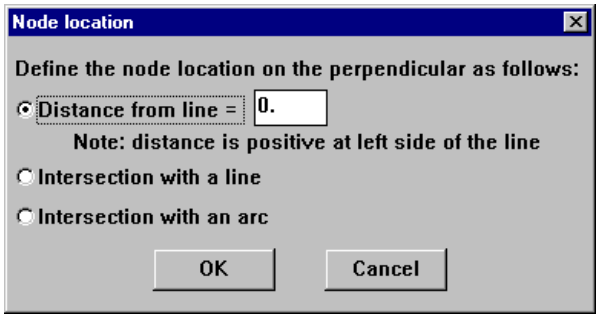
◉ **from node**

select nodes **1**, **2** and **3**; the new node is created at the perpendicular from node **3** to the line joining nodes **1** and **2**.

◉ **from nodes**

select nodes **1** and **2**, then select a series of nodes using the standard node selection option; the new node are created at the perpendicular from these nodes to the line joining nodes **1** and **2**.

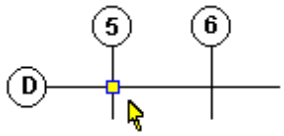
from line
select nodes **1** and **2**; the new node is on the line perpendicular to line **1-2**, drawn from node **1**, according to one of the following options:


Node location
Define the node location on the perpendicular as follows:
Distance from line =
0.
Note: distance is positive at left side of the line
Intersection with a line
Intersection with an arc
OK
Cancel

- at a specified distance from node **1**. Note that a positive value is to the left when standing at **1** and looking towards **2** (the example above shows a negative value)
- at the intersection with another line (defined by two additional nodes - **3** and **4** in the example above)
- at the intersection with an arc (defined by three additional nodes)

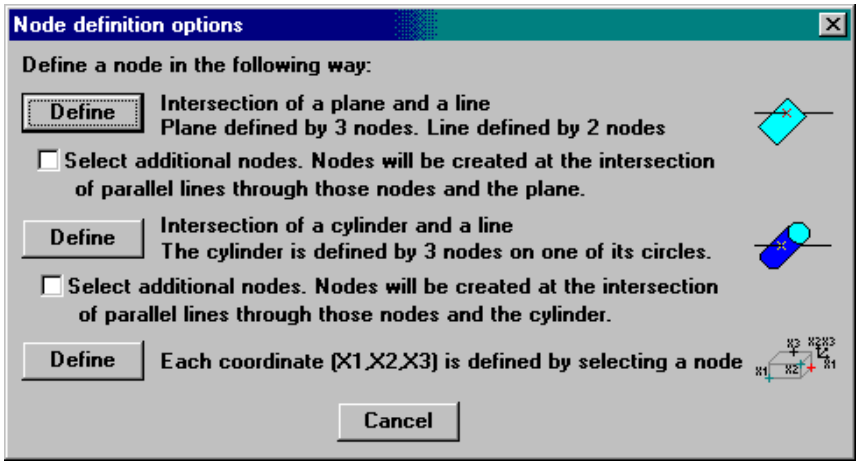
- DXF points
  This option is displayed only if a DXF drawing has been added to the display background
  ☑ ONLY the end points of DXF lines may be selected as node locations
  ☑ the DXF end points will not be highlighted with the ■

- Grid lines
  Define a node at the intersection point of any two grid lines. For example:

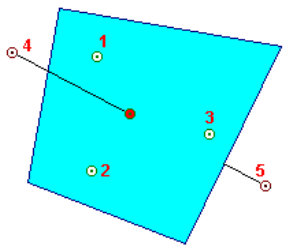


Note that this option not available when a background DXF drawing is displayed.

- More


Node definition options
Define a node in the following way:
Define
Intersection of a plane and a line
Plane defined by 3 nodes. Line defined by 2 nodes
Select additional nodes. Nodes will be created at the intersection of parallel lines through those nodes and the plane.
Define
Intersection of a cylinder and a line
The cylinder is defined by 3 nodes on one of its circles.
Select additional nodes. Nodes will be created at the intersection of parallel lines through those nodes and the cylinder.
Define
Each coordinate (X1,X2,X3) is defined by selecting a node
Cancel

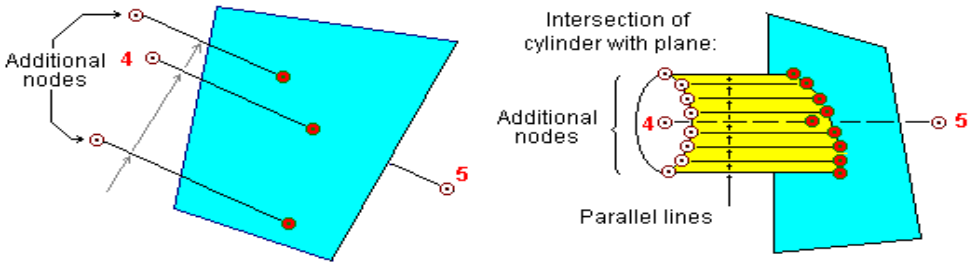
- Intersection of plane & line



- Select any three existing nodes that define the plane
- Select any two existing nodes that define the line

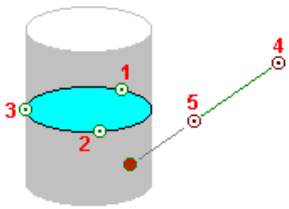
The five nodes cannot lie on the same plane.

☑ **Select additional nodes**
Select a series of nodes; the program creates nodes at the intersection of the plane and lines that are parallel to the original line but pass through the additional nodes. For example:



- Intersection of cylinder & line

  The new node is created at the intersection of a cylinder and a line:
  - Select any three existing nodes that define the a circle on the cylinder (the program assumes that the cylinder extends infinitely in both directions)
  - Select any two existing nodes that define the line.

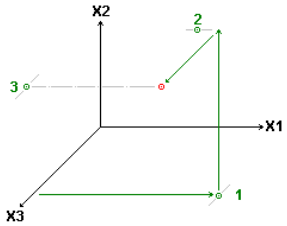


Note:
- There are two intersection points; the new node will be created at the point closest to the line end node.

☑ **Select additional nodes**
Select a series of nodes; the program creates nodes at the intersection of the cylinder and lines that are parallel to the original line but pass through the additional nodes.

- Node - 3 existing nodes

  Select three nodes:
  - X1 of the first node = X1 of the new node
  - X2 of the second node = X2 of the new node
  - X3 of the third node = X3 of the new node

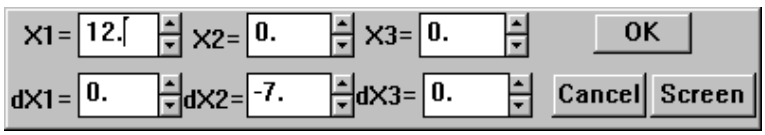


2.1.2 Line of Nodes - Equal Spacing

Use this option to define a line of nodes where the spacing between them is equal.

To define the line:
- define the start node of the line as explained in 2.1.1 - Single node
- define the end node location:

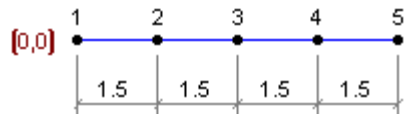
  The dialog box at the bottom of the screen is:



  where:
  X1,X2,X3 = location on the screen (global coordinates).
  dX1, dX2, dX3 = distance **from the start node**.

  Move the until the correct coordinates are displayed or type in the correct values in the text boxes.
- Specify the number of segments. For 'n' nodes (including the end nodes), there are 'n-1' segments.

Example:



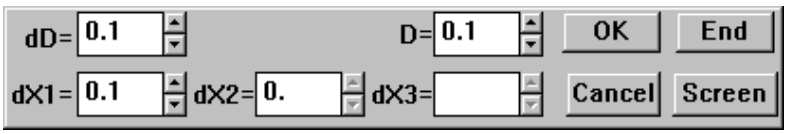
- move to: X1 = 0.0 X2 = 0.0 ; click the mouse.
- move to: X1 = 6.0 X2 = 0.0 ; click the mouse.
- specify four segments

Additional options (refer to 2.1.1):
- Define with same coordinates as existing node.
- Define end node at intersection of lines.
- Define at the center point of a line between two exist nodes
- Define at the end pont of a line in a DXF background drawing

2.1.3 Line of Nodes - Unequal Spacing

Use this option to define a line of nodes where the spacing between them is **not** equal.
- Define the start and end nodes of the line as explained in 2.1.2 - Line - Equal.
- Point to the intermediate nodes along the line, as follows:
  The moves along the line joining the start and end nodes.

  The dialog box at the bottom of the screen is:



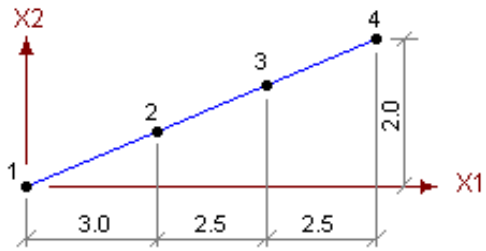
  where:

  dD = distance **from the last node defined on the line** as measured along the line.
  dX1,dX2,dX3 = distance **from the last node defined on the line** (global coordinates).
  D = distance **from start node** as measured along the line.

  Note that dX1 or dX2 (the closest to the line) and not dD is updated at the Step rate, a feature that is very convenient when only the projected spacings of a diagonal line on a global axis are known.

  In many cases it may be more convenient to type the spacing directly in the text boxes rather than by continuously adjusting the Step for each node.

Example:



- move the to the location of Node 1 and click the mouse.
- move the to the location of node 4 - dX1 = 8.0, dX2 = 2.0 and click the mouse.
- move the along the line until dX1 = 3.0 is displayed; click the mouse; node 2 is created.
- move the along the line until dX1 = 2.5 is displayed; click the mouse; node 3 is created.
- move the outside the line and click the mouse.

Additional options (refer to 2.1.1):
- Define with same coordinates as existing node.
- Define end node at intersection of lines.
- Define at the center point of a line between two exist nodes
- Define at the end pont of a line in a DXF background drawing.

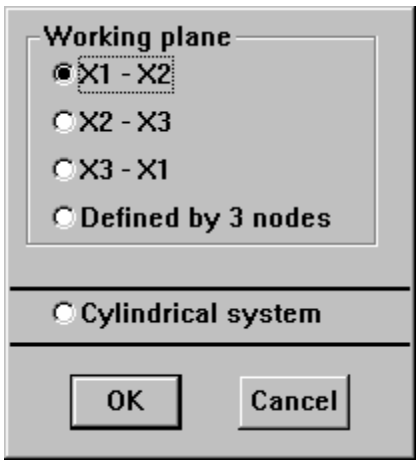
2.1.4 Nodes - Coordinate system

By default the moves on the X1-X2 plane (even if the display is rotated) and node coordinates are defined relative to this Cartesian global system.

This option allows you to define coordinates relative to a another system.

A different Cartesian system or a cylindrical system may be defined:
- the will move along the plane of the Cartesian system (referred to as the "Working Plane") or along the arc of the cylindrical system.
- node coordinates are defined relative to the axes of the new system.

There are two options available:



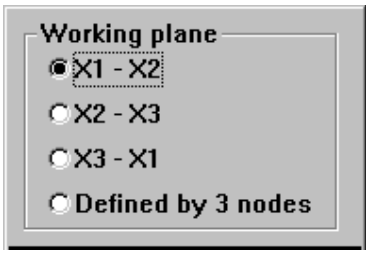
- Working Plane
  Define the working plane as a different global plane, or as a plane formed by any three nodes in the model. Refer to 2.1.4.1.

- Cylindrical System
  Define a cylindrical coordinate system. The crosshair movement is controlled by defining the angle, radius and height from an arbitrary origin. Refer to 2.1.4.2.

2.1.4.1 System - Working Plane

The working plane is a plane in space along which the moves. By default, X1-X2 is the working plane. Any of the three global planes or any arbitrary plane may be defined as the current working plane.

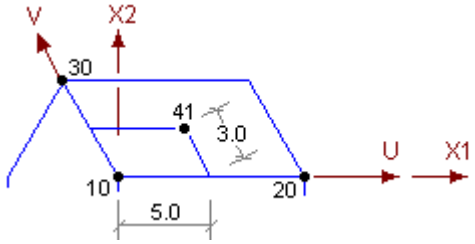
If the working plane is an arbitrary plane **not** parallel to one of the global planes (defined by three nodes), the coordinates displayed in the Data Options Area are "U, V and W", where:



The origin is located at the first node selected to define the working plane.
- U is measured from the first node in the direction of the second node. The value of U is revised by moving the mouse horizontally.
- V is measured perpendicular to U in the general direction of the third node. The value of V is revised by moving the mouse vertically.
- W is the dimension perpendicular to the plane. Its positive direction is determined by the right-hand rule.
- The value of W is revised in the Dialog Box at the bottom of the screen.

Note:
- the Working Plane option is in effect only for node definition.
- the program will automatically rotate the model if the working plane is perpendicular (or nearly perpendicular) to the screen when you enter the node definition option.

Example:
Use a working plane to define node 41 on the sloped roof below. The node location relative to the plane of the roof is known (as detailed in the figure) but not relative to the global coordinate system.



- define a Working plane as **Defined by 3 nodes**.
- select nodes 10, 20 and 30 **in that order** to create the Working Plane U-V-W as shown.
- select **Single node**; move the until U = 5.0 and V = 3.0 are displayed in the Data Options Dialog Box (note that the moves along the working plane only).

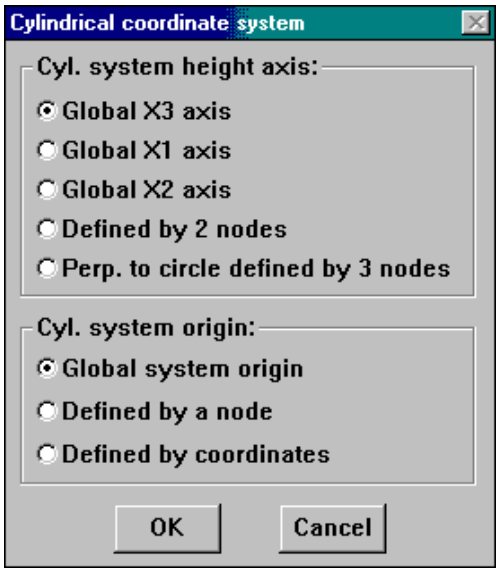
2.1.4.2 Cylindrical Coordinate System

Coordinates may also be defined relative to a cylindrical coordinate system. If a cylindrical system is specified, the coordinates used to define node locations are:
- radius (R) instead of X1
- angle (Ang) instead of X2
- height (H) instead of X3.

A cylindrical coordinate system has a central non-cylindrical 'height axis' which may point in any direction and which passes through a selected 'origin' node.

To cancel a Cylindrical System, define a Working Plane.

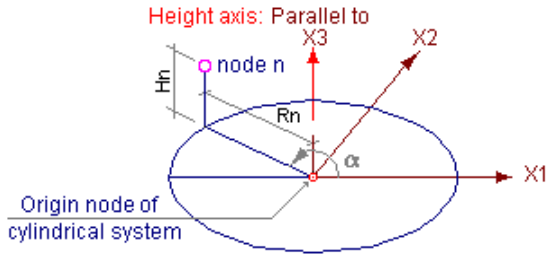
Select the global non-cylindrical axis and its location:



The origin location depends on the option used to define the height axis:

Height axis	System origin
Parallel to global axis:	according to option selected in menu
Defined by 2 nodes:	at first node selected
Perp. to circle:	at center of circle

For example, if X3 is chosen as the "height axis" and the origin is set at any arbitrary location, the cylindrical coordinate system is:



The options **Line equal, Line general** and **Grid** may all be used while a cylindrical coordinate system is in effect.

** Blank page **

2.2 Restraints

Define the location of restrained nodes, i.e. support locations, by specifying which nodes have restrained degrees-of-freedom. Each restrained degree-of-freedom prevents translation or rotation in a specific direction. Models without any restrained nodes are unstable (i.e. the stiffness matrix is singular).

The restraints are normally defined in the Global Coordinate System directions. In the case of a sloped support, etc., restraints may be defined relative to any arbitrary local axis.

Support locations are specified by moving the crosshair on the graphic display (using the mouse or the arrow keys) to point to an existing node.

Select one of the following options:

All relevant translation and rotation global degrees-of-freedom are restrained.

All relevant translation global degrees-of-freedom are restrained.

Any other combination of restrained global degrees-of-freedom..

Define a 'local' support coordinate system (a system for supports not parallel to global axes) and assign supports to these systems.

Connect selected nodes by means of "Rigid links" (Master-slave nodes).

Delete supports.

2.3 Beams

Define beam elements by specifying

location : designate the end nodes.
properties : define properties (A,I), dimensions or a steel shape
material : select a program material or define a new one
local axes : specify the local x2/x3 axis directions
orientation : specify major/minor axes orientation (relative to the local axes)
releases : define pinned connections
rigid offsets : define rigid segments at the beam ends

Select one of the following options:

- Define one beam element by identifying its end nodes, or -
- Define a series of beams all lying on a straight line. The beams are defined by identifying the nodes at the start and the end of the line; the program automatically locates all intermediate nodes and connects them with beams, or -
- Define a continuous string of beams, where the start node of any beam is the same as the end node of the previous beam.

Define a parallelogram grid of beam elements. The grid is defined by identifying the three corner nodes on the 'base' line and the 'height' line of the grid; the program automatically searches for all intermediate nodes and creates a grid of beams.

Define a series of beams that all start on a common line and end on a different common line.

Delete a beam already defined.

Define section properties (including material) and assign them to beams.

Define pinned end connections.

Define rigid offsets at the ends of beam elements and assign them to the beams.

Renumber a beam element already defined.

Divide an existing beam into two or more beams if intermediate nodes are located along the length of the beam.

Specify or revise the direction of the local axes for beams.

When a stage other than **Whole model** is the current stage:
- beams may be ‘removed’ or ‘restored’
- Different properties, releases and offsets may be defined

2.4 Elements

Define quadrilateral or triangular elements by specifying:

- location : designate the end nodes.
- properties : define the element thickness
- material : elements may be isotropic or orthotropic
- local axes : specify the local x3 axis direction

When "Elements" are selected in the geometry Main Menu, the following options are available:

Define one triangular element by identifying its end nodes.

Define one quadrilateral element by identifying its end nodes.

Define a chain of triangular elements where only the third node of successive elements need be defined; the program uses two nodes of the previous element (common to both elements) to complete the new triangle.

Define a chain of quadrilateral elements where only the third and fourth nodes of successive elements need be defined; the program uses two nodes of the previous element (common to both elements) to complete the new quadrilateral.

define a 'base' line consisting of a chain of nodes and a 'height' line, also a chain of nodes, that starts at one end of the base line. The program copies the base line to every level on the height line, generating nodes and connecting them with elements. If the base line is a semi-circle and the height line is a perpendicular line, the program will generate a half cylinder; if the second base line is a semi-circle with a smaller radius, the surface will be conical. This option can also generate plane grids

A powerful command for automatic element generation. Generate a grid of nodes along with the corresponding grid of elements. The grid outline is defined by specifying a contour, and the size of the generated elements is determined by user defined parameters.

Delete elements already defined.

Renumber elements already defined.

Define section properties (including material) and assign them to finite elements.

Revise the direction of the local x3 axis. This option is used to reverse the direction of the local x3 axis as set by default by the program.

to define the edges of bending elements as "pinned".

define offsets perpendicular to the element.

The following options are available when a stage other than **Whole model** is the current stage:

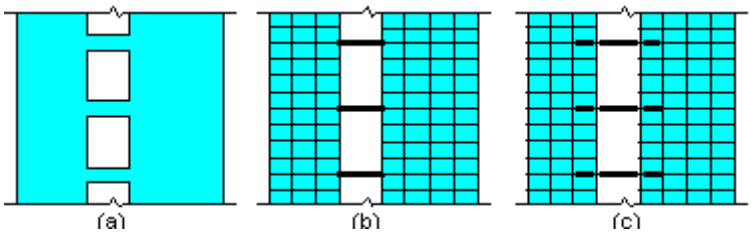
Define/select construction ‘stages’. Different properties may be defined for each stage and beams/elements may be removed. New beams cannot be defined when a stage other than Whole model is active

“Remove” an element from the current stage. Select beams using the standard element selection options. Inactive elements are not displayed.

Restore an element to the current stage.

Note:

- Models may contain both beam elements and finite elements.
- the end nodes of **plane strain** finite elements do not transfer bending moments to adjacent elements, i.e. the corners of these elements are all 'released'. Referring to the shear wall in Figure (a), if the lintel beams are modeled by beam elements as shown in Figure (b), no bending moments will be generated in them. The beams should be extended into the wall as shown in Figure (c).



2.5 Springs

Define linearly elastic spring supports - translational and rotational - at nodes.

When an elastic support is defined in a specific direction at a node, the node must be unrestrained in that direction.

To define or revise springs.

To delete springs.

To write the spring constant values on the graphic display (for defined springs).

Define translational springs that act in the positive or negative direction only of the selected axis, but not both.

Define spring constant per area/length and then select elements/nodes defining area/length; program will automatically calculate spring constants for relevant nodes.

To revise a spring, select **Define**, select the node and enter the revised value.

Note:
To define a translational spring in a direction **not** parallel to a global direction, assign a local restraint system to the node, as explained in Springs - local. **Do not define the global components of the spring constants using S sin(θ) and S cos(θ)**.

2.6 Copy

Copy is a powerful option which enables the user to copy a block from the model - including nodes, elements, properties and releases - to a different location.

All elements in the copied block are drawn parallel to the original ones. The location of the copied block is at a specified distance from the original.

The copied block may also be rotated with respect to the original block. The program may stretch or shrink the dimensions of the block, but will always maintain the same node/element layout.

Create a mirror image of the original block about an axis of symmetry.

Note:

- more than one copy of the block may be created with one command
- the program will not generate a new node at the location of an existing node but will use the existing node when creating the elements in the new block.
- the program will not generate a new element at the identical location of an existing element.
- the copied element will be assigned with the property group number of the original element.
- Releases are automatically copied.
- Beam local axes (Copy+rotate/Mirror)
  The program tries to rotate the local axes along with the rotation of the block. For example: a beam with x2 pointing to the centre of a circle is copied radially around the circle; the x2 axis of all the copies of the beam will also point towards the centre of the circle.
- Element local axes
  The local coordinate systems of the copied elements are selected so that the axes of the copied elements point in the directions that are as near as possible to the directions of the axes of the original elements.

The block to be copied is defined by selecting a group of nodes using the standard Node Selection option; only elements with all of their end nodes selected will be included in the block. The location of the copied block is defined by entering the new location of reference nodes (at an existing node or at any coordinate); if the distance between the reference nodes is changed in the copied block, then the element dimensions will be revised proportionally.

- the location of the new block may be defined by rotating and translating the original block.
- the two blocks may have a common intersection line; the program automatically Unifies the two blocks so that they are connected.
- the program may be instructed to connect the new nodes and the corresponding original nodes with beams.
- the command may stretch or shrink the dimensions of the block, but will always maintain the same node/element layout.

For all options, select the nodes that define the block.

2.7 Solid elements

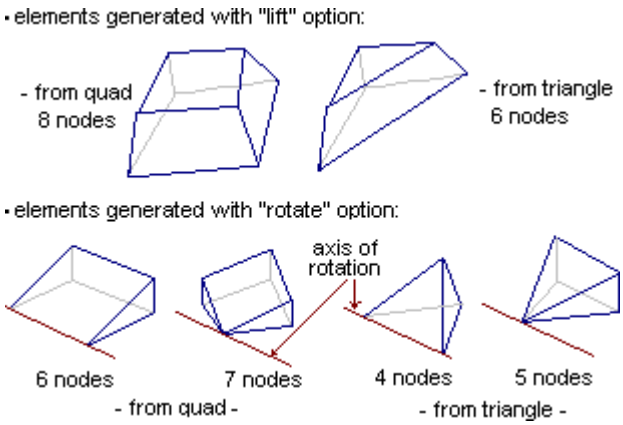
Define solid finite elements.

Note: **the Bridge postprocessor cannot solve models that include solid elements**.

Solid elements are stress elements with actual thickness defined by the distance between end nodes.
The results are stresses and principal stresses at the corner nodes.

The elements are created from existing quadrilateral or triangular plate elements by lifting or rotating the plate element to the nodes on the opposite face. The plate elements used to generate the solid elements may then be erased.

The generated elements may have 4,5,6,7 or 8 nodes:



Note that solid element numbering is independent of beam/plate element numbering.

Generate solid elements by "lifting" existing plate elements.

Generate solid elements by "rotating" existing plate elements.

Delete existing solid elements.

Renumber existing solid elements.

Define the material properties for solid elements.

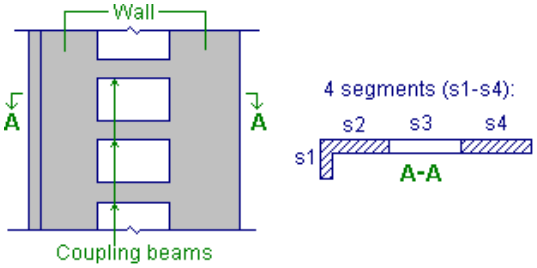
Define the material properties for solid elements.

When a stage other than **Whole model** is the current stage:
- elements may be 'removed' or 'restored'
- properties may be modified.

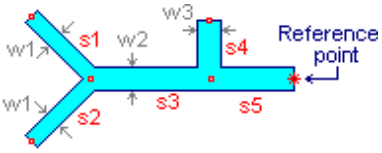
2.8 Wall Elements

The Wall option enables the quick definition of complex walls that extend across multiple levels in the model.

- the wall cross-section is defined first and then is attached to the model by selecting a vertical line of nodes; the program automatically creates any additional nodes that are required
- each wall section may consist of multiple segments together with coupling beams, for example:

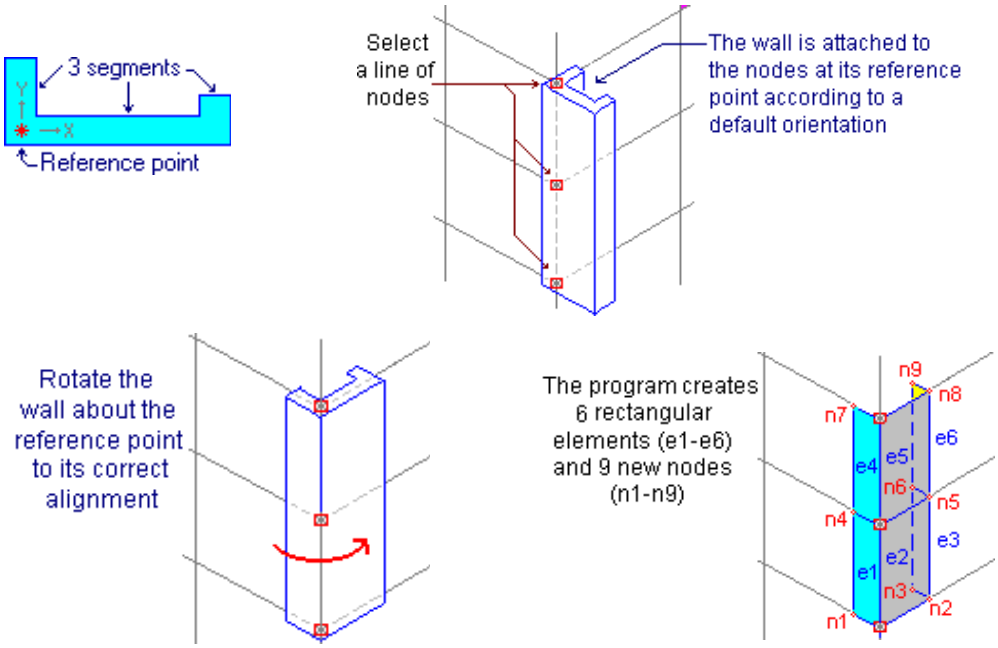


- the number of segments is unlimited, the arrangement and orientation of the segments is not restricted and each segment may have a different width (coupling beams may have varying heights). For example:

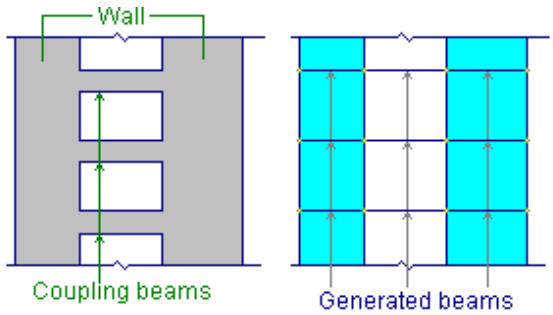


- each wall has a "reference point" at the end of one segment. This point is used to attach the wall to nodes in the model (referred to as the "Attachment nodes").
- when the wall is attached to the model, the program generates a series of rectangular elements from the segments and creates the necessary nodes at the corners.

Example: the following wall is defined and attached to the model:

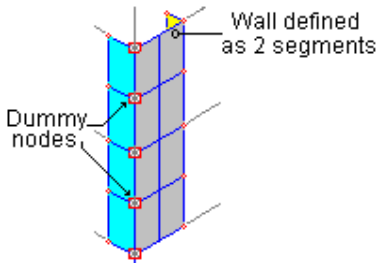


Each coupling beam generates three beams in the model: the coupling beam itself and a beam in each of the adjacent walls that allows transfer of moments into the coupling beam (elements cannot transfer moments at their corners!): For example:

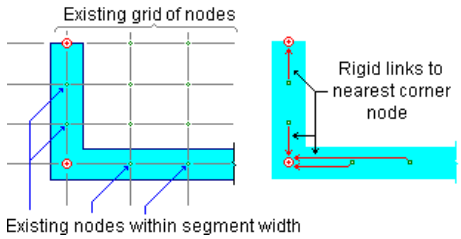


Note:

- The program creates a single element between nodes along the height axis for each segment. Normally this implies a single element per story height. Testing has shown that the results are sufficiently accurate for typical long multi-storey walls (refer to the *STRAP* Verification manual). For short walls, improved accuracy will be obtained if dummy nodes are defined between floor levels and long segments are defined as more than one segment.



- The wall segments are connected to the rest of the model only at their end nodes, however there may be existing nodes that lie within the segment width but are not connected to the wall. The program can search for such nodes and automatically create rigid links to the nearest wall corner node.



- The results for wall elements are presented in the form of beam results - one axial force, moment and shear value for each generated element.

When a stage other than **Whole model** is the current stage:

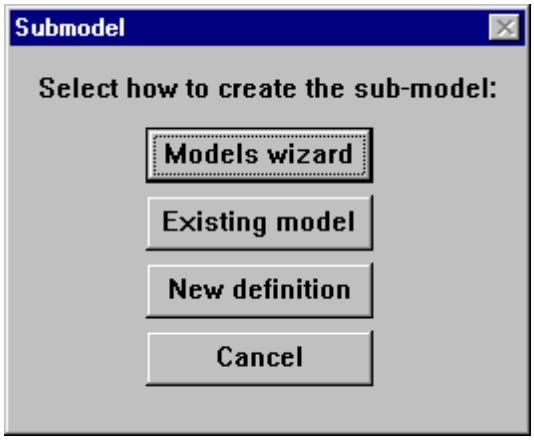
- walls may be ‘removed’ or ‘restored’
- properties may be modified.

2.9 Sub-model

A sub-model is a model defined in a temporary working area. The model may then be copied to the current model any number of times, at any location and at any angle. Note that the entire sub-model geometry may be copied to the main model, including supports, springs, offsets, rigid links, etc.

The sub-model may also be saved as a separate model in any directory.

There are three methods for creating the sub-model:



Note that the main model and the sub-model are open simultaneously until the sub-model is closed; the user may jump back-and-forth at any time by clicking the icons.

The following options are available:

Models Wizard:

Create the sub-model from one of the models in the Wizard library.
Note that the sub-model generated by the Wizard may be revised before it is added to the current model.

Existing model:

Retrieve an existing model from any working directory.

Note that the sub-model may be revised before it is added to the current model.

New definition:

Define a new model and add to the current model.

Note:
- the program will not create a new node at the location of an existing node.
- supports/springs at nodes in the sub-model that are not copied will not be added to the main model.
- the program will not create a new beam/element at the identical location of an existing beam/element.
- restraints or springs defined parallel to global axes in the sub-model will be oriented in the closest possible global direction in the main model. Restraints defined according to a local system will be rotated to the same orientation relative to the sub-model.

2.10 Stages

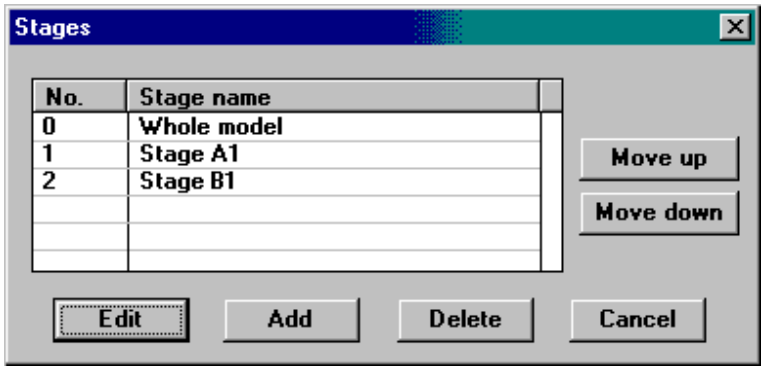
Define construction 'stages' for the current model:

- the default stage is the **Whole model**; it represents the final structure and contains all elements and the properties and supports/springs present at the end of construction.
- all other stages represent intermediate steps during construction; beams and elements may be removed or restored (but cannot be created), different properties and supports may be assigned, and only specified load cases may be applied.

The program treats each stage as a separate model and calculates the results based on the stage geometry and the applied load cases.

The stages are defined and activated in this option; elements, properties, restraints, etc are applied/removed from the stages in the various Geometry and Load options.

For example:



*** Blank page ***

3 Loads

- The program can solve the model for multiple load cases. Each load case may consist of joint loads, beam loads, element pressures and support displacements.
- Load cases may be combined to form a new load case.
- Beam and element loads may also be defined relative to the global coordinate system; standard load patterns may be stored in a file and recalled.
- Each load case may be assigned to a different geometry 'stage'.
- All defined loads for the current load case are displayed graphically superimposed on the geometry.

Select one of the following options:

To begin the definition of a new loading case.

To revise the loads in an **existing** load case.

To delete an existing load case.

Deactivated load cases are not solved but are not erased.

Automatically generate a series of "moving" load cases from a single basic case. The global loads in the basic case will be offset by a constant increment in each successive generated case.

Automatically generate a series of load cases with alternating patterns of live load from basic load cases containing the dead and live loads on all spans. The patterns are arranged according to Code requirements for calculating maximum and minimum moments in beams.

Calculate second-order (P-D) forces and moments.

Define wind load cases according to Code requirements.

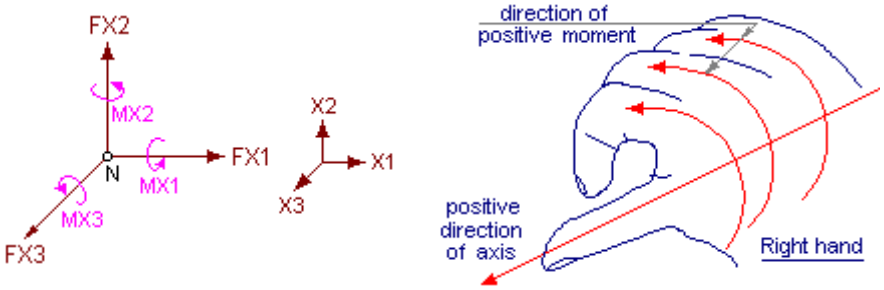
Copy an entire load case.

Define sway unit load cases at specified nodes. These load cases are required for the sway/drift control option in the Steel postprocessor.

3.1 Joint loads

Joint loads are defined relative to the global coordinate system or to any arbitrary local system defined by three nodes.

The positive force and moment sign conventions are:



3.2 Beam loads

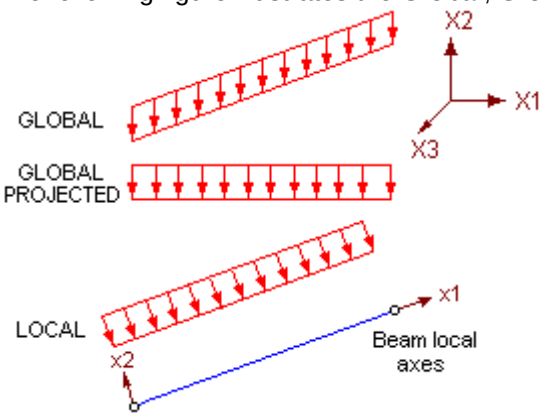
Beam loads are uniform, linearly distributed or concentrated loads applied anywhere along the length of the beam between the end nodes.

Self-weight and temperature (expansion / contraction or gradient) are defined as Beam Loads.

Beam loads may be defined either in a direction parallel to one of the global coordinate axes or parallel to one of the local coordinate axes of the beam.

Local : the load is parallel to the beam local axis specified.
Global : the load is parallel to the global axis specified and is applied to the total beam length.
Global projected : the load is parallel to the global axis specified, but is applied to the projected beam length as projected onto the specified global axis (uniform and linear loads only).

The following figure illustrates the **Global**, **Global projected** and **Local** beam loads commands.



3.3 Element loads

Element pressures are applied to the entire surface area of quadrilateral or triangular elements.

The pressure is not necessarily applied normal to the element surface; the load can be applied in any of the local or global axis directions. In all cases, the total load applied is the pressure multiplied by the surface area of the element.

3.4 Support displacement

Support displacements may be entered in the direction of any degree-of-freedom, including rotation.

Note that the nodes must be restrained in the same degrees-of-freedom as the defined d displacements.

3.5 Combine loads

Use this option to combine existing loading cases to create a new load case; the existing load cases may be multiplied by a factor.

Note that the ***STRAP*** result module also has an option for combining load cases. In general, it is more convenient to define combinations after the solution rather than at this stage, because:

- the solution time is decreased
- combinations may be revised without solving the model again
- since the P-Delta effect is non-linear, the rules of superposition do not apply. Therefore, load combinations for models with P-Delta must be defined here in LOADS, rather than in the results module.

This option may also be used to insert an existing loading case into a new loading case.

3.6 Global loads

Load locations may be defined relative to the global coordinate system. The program locates the nodes and elements surrounding the "global loads" and converts the loads to equivalent joint loads or element loads, as requested by the user.

This option is useful in models characterized by load patterns which do not coincide exactly with the nodes or elements, such as bridges.

Global loads can be entered directly or may be stored and recalled from a file.

3.7 Solid loads

Define temperature loads or self-weight for solid elements. All other loads (pressure, linear loads, concentrated loads, etc) must be defined by applying them to beams or plate elements (dummy or regular) defined parallel to the solid elements surfaces.

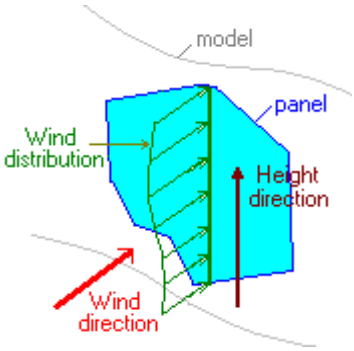
3.8 Wind loads

This option generates wind loads according to Code requirements.

The user defines:

- the outline of areas referred to as "panels", or selects beams and defines the perpendicular panel width.
- the direction in which the wind acts.
- the "height" direction of the panel.
- various Code parameters.

The program calculates the wind pressure distribution on the panel according to the Code and distributes the load to the nodes/beams/elements in the panel area.

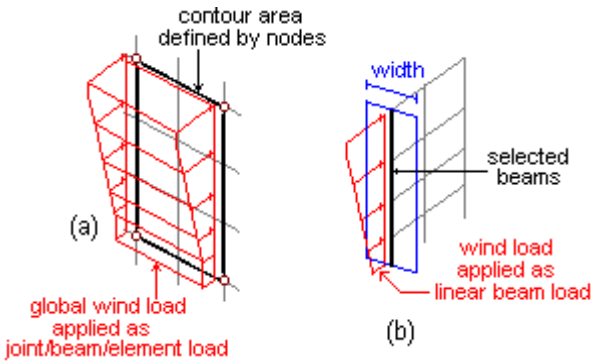


In general, each wall or roof plane should be defined as a separate panel.

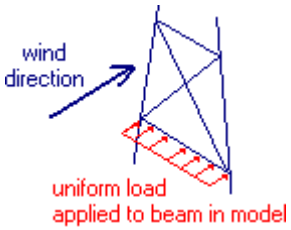
Note:
- Wind pressures are applied to the nodes/beams/elements using the same methods that are used to apply global area loads.
- Dynamic effects are not considered.

The wind loads may be generated by specifying:

- a contour area: the global area loads calculated by the program are applied to either nodes, beams or elements.
- a series of beams and the perpendicular width: the program applies linear beam loads (this is the only option available for plane frames)



- as uniform beam loads on a "lattice" structure. the program assumes that the model is an open structure, i.e. the wind blows through the model. This option is primarily intended for transmission towers, open trusses in industrial buildings, etc.



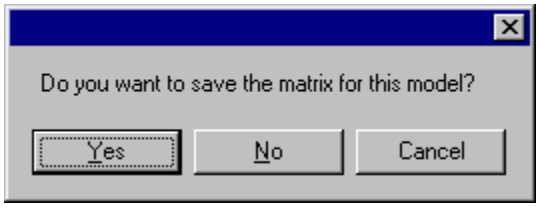
The program calculates the projected surface area of each member perpendicular to the wind direction, calculates the wind load for individual members in a lattice structure according to the Code and applies it to the selected beams.

4 Solve the model

4.1 Solve - general

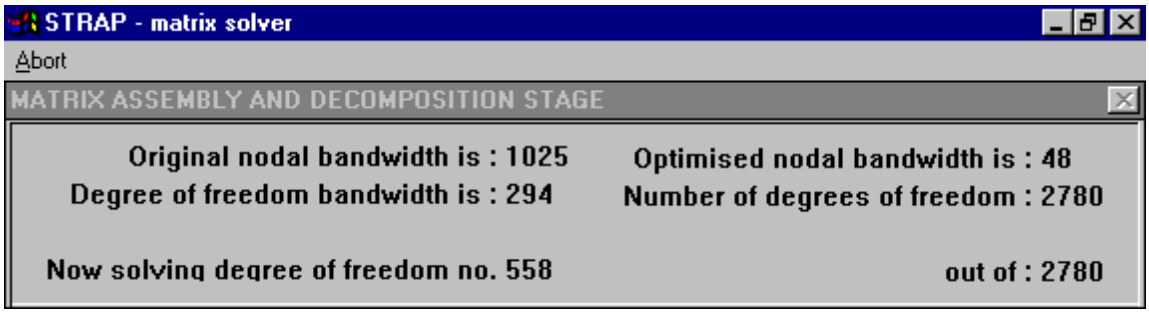
Refer to Solution method (4.2) for a detailed explanation on the method used by the program to solve the model.

The program deletes the inverted stiffness matrix at the end of the solution phase in order to save disk space. However, for large models it may be advisable to save the stiffness matrix if the same geometry will be solved with different loads; the solution will then skip the matrix inversion.



Press to save the matrix.

At the start of the solution, the program displays the header:


STRAP - matrix solver
Abort
MATRIX ASSEMBLY AND DECOMPOSITION STAGE
Original nodal bandwidth is : 1025
Optimised nodal bandwidth is : 48
Degree of freedom bandwidth is : 294
Number of degrees of freedom : 2780
Now solving degree of freedom no. 558
out of : 2780

The original and optimized nodal bandwidths appear on the second line. The actual degree-of-freedom bandwidth and the total number of degrees-of-freedom appear in the third line of the table.

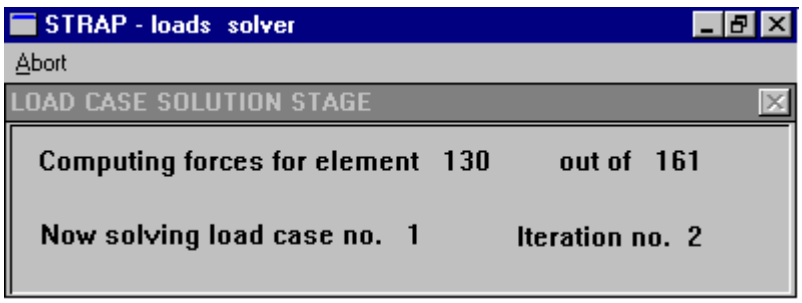
The program displays the rate of progress of the solution: Estimate the solution time by the rate-of-change of the number of degrees-of-freedom already solved.

If the model is unstable, the program will display singularity messages. The messages may be printed.

Refer to:

4.3 - Singularity
4.4 - General problems

After solving all d.o.fs, the program will solve the load cases:



The iteration number appears only if non-linear options (P-Delta, unidirectional springs, etc.) were specified.

The program copies a back-up of the solution to the disk every few minutes (refer to Setup). The solution may be stopped at any stage by clicking **Abort**. After a few seconds the program will return to the operating system. The solution can later be continued from the point of the last backup as follows:

- start the program
- select the current model
- select **Solve** on the menu bar

To restart the solution from the point of interruption, select **Resume solution** when the following menu is displayed:

4.2 Solution method

The program solves the model by the stiffness method. This method solves models on the basis of joint equilibrium equations in terms of stiffness coefficients and unknown joint displacements. The method formulates the matrix equation

[K] * {d} = {P}

where:

[K] = the stiffness matrix and is a function of the geometry
{d} = the nodal displacement vector
{P} = the applied nodal forces vector

The equation is solved for the unknown node displacements and hence the internal element forces or stresses.

The stiffness matrix is a square matrix where each side is approximately ndof*nj (where ndof is the number of degrees-of-freedom per joint and nj is the number of joints in the structure), and is symmetric about the diagonal starting in the upper-left corner of the matrix.

Most of the values in the matrix are equal to zero; The non-zero values are usually clustered along the diagonal, forming a diagonal band. The width of this band is called the 'bandwidth'.

The significance of the bandwidth is that its size is the most important factor in determining the solution time. The bandwidth size is dependent on the way the nodes and elements in the model are numbered. For example, numbering the nodes in a multi-story model vertically instead of horizontally will increase the bandwidth size.

However the user need not worry about the numbering method. The program **automatically** renumbers the model so that the bandwidth has the minimum width and so minimizes the solution time. The renumbering is internal, so the results always appear according to the defined node and element numbers.

The program begins the solution by inverting the stiffness matrix. The stiffness matrix is a function of the geometry only and the inversion of the matrix takes up the majority of the solution time.

If the program finds a zero value along the diagonal when building the stiffness matrix, the program replaces it with the value 1.0E+20, and displays a ZERO STIFFNESS warning.

The program decomposes the matrix by the CHOLESKY method.

During the decomposition of the matrix, if the program discovers a value X along the diagonal that after decomposition has a value 1.0E-9*X, the program replaces it with the value 1.0E-7*X and displays a SINGULARITY message.

Beam Elements:

The program includes the shear contribution in the beam stiffness matrix and allows for a reduced shear area.

Triangular Elements:

- Bending: The program uses the HSM element (Hybrid Stress Model) as described in:
  A Study of 3-Node Triangular Plate Bending Elements
  Jean-Louis Batos
  International Jour. for Numerical Methods in Engineering
  Vol. 15 1771-1812 (1980)

- Plane Stresses: The program uses a constant strain plane stress element.

Quadrilateral Elements:

- Bending:
  The program divides each rectangular element into four HSM triangular elements by creating a new node at the element centre. The program then uses matrix condensation to delete the new node.

- Plane Stresses: The program uses a linear strain plane stress element.

4.3 Singularity

The program may also display additional messages during the solution phase indicating that it encountered problems in solving the equation i.e. the stiffness matrix was found to be singular.

Singularity warnings may be printed.

Singularity indicates that the matrix determinant equals zero. The cause of singularity is instability of the model.

ALWAYS DETERMINE the REASON for SINGULARITY

There are two types of singularity:

Local Singularity:

The model as a whole is stable but there is local singularity at a node:

zero stiffness at node _ in dof _ . dof was restrained

The program will restrain the DOF and proceed with the solution.

Several examples are:
- Plane truss : X3 was not restrained at all nodes.
- Plane frame : All beams connected to a node are pinned.
- Space frame : Torsional moment of inertia (J) not defined.

Note:
In many cases of local singularity the corrective action taken by the program will lead to the correct solution (e.g. plane truss example above). However, the solution time required will be greater.

Structural Singularity:

The entire model is unstable. The following message appears:

Singularity or near singularity detected at node __

The program will take corrective action, i.e., supply missing restraints, in order to proceed with the solution.

Note that the program arbitrarily restrains the first joint in the model that will make the model stable. The corrected model usually does not correspond to the model that you intended to solve; check the reactions in the result table to see if the program created new supports or display graphic displacements and check if the deflected structure has the correct form.

If the model is not as intended, ignore the results and return to geometry to revise the input.

Note:
- this singularity message may be displayed when there is a large difference between maximum and minimum moment-of-inertia values defined for property groups.

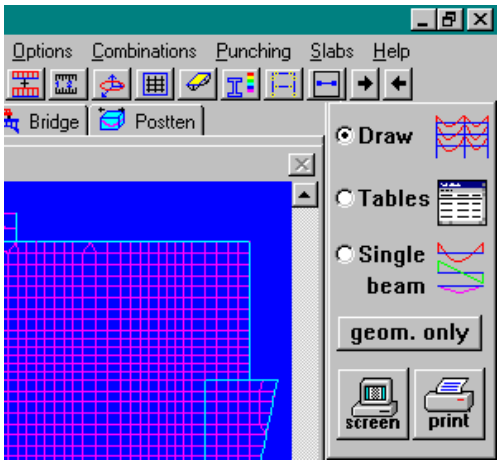
4.4 Problems - General

- If the solution is interrupted by a power failure or computer malfunction, the solution may be restarted from the point of the last backup as explained in Solution time / backup.
  - Warning messages (solution will be completed):
  - if the model consists of two or more unconnected parts.
  - if two or more beams connect the same two nodes
  - too many beams connected to one node (node/element optimization is discontinued).

- Error messages (solution is aborted):
  - Total number of degrees-of-freedom exceeds 192,000 (32,000 x 6).
  - Various errors in the geometry.
  - Not enough space on disk -
  - At the start of the solution, the program checks if sufficient disk space is available to complete the calculations. If not, the program displays a warning message and lists the number of bytes required. Erase unnecessary files from your disk and restart the solution.

5 Results

The program enters the output module of the program after completing the solution.



To display graphic/tabular results:

- Define the load **Combinations** (menu bar). Refer to 5.1.
- Specify **Options**, e.g. result axes (menu bar). Refer to 5.2.
- Select - ◉ **Tables** or ◉ **Draw** or ◉ **Single beam**
- Click:

to display the tabular/graphic results.

to print the tabular/graphic results. Refer to 1.9.8 and 1.9.9 for standard print options.

- **geom. only**

  redraw the current screen with geometry only and without results.

  This option is convenient when rotating large models with results that take a considerable amount of time to calculate and draw, e.g. contour maps. The program must recalculate the results every time one of the rotate icon is clicked.

  In such cases, select **geom. only**, rotate the model (without the results), then select ◉ **Draw** to redisplay the results.

Note:

- Results from several different models can be combined prior to displaying the results. Refer to 1.5.4 - Combine results of 2 projects.
- Tables may be written to ASCII files in user-specified format using the *STBatch* utility.
- The following terminology is used throughout the output modules:

Load **CASES**	Load cases defined in “Loads” (prior to solving the model). Note that some of these cases may be in fact “Combined load cases” but are still referred to as CASES.
Load **COMBINATIONS**	Combinations of load cases defined after solving the model using the “Combinations” option

5.1 Combinations

Use this option to define combinations of the load cases for the following output modules:
- tabular results
- graphical results
- design postprocessors

Note:
- For a detailed explanation of "Groups" and "Library" , refer to 5.1.1 - Combinations - General
- the rules of superposition do not apply for non-linear elements. Therefore, load combinations for models with tension/compression only elements, unidirectional springs, etc, must be defined in loading ("Combine ld") and not after the solution using this option.

5.1.1 Combinations - General

- Each "Combination" is defined as a combination of load cases, each of which may be multiplied by a factor.
- Combinations may also be retrieved from a "Combination Library"; combinations defined for the current model may be added to the combination library for use in other models.
- "Groups" of load cases may also be defined. If a group is added to a combination definition, the program will either:
  - automatically generate a separate combination for each load in the group, or
  - add the sum of the load cases in the group to the combination.

Examples:

- Groups:
  The following load combination is required:

  1.4*Dead + 1.6*Imposed + 1.6*Crane

  where there are 5 different load cases with Crane loads, each corresponding to a different point of application of the load.

  Instead of defining 5 separate combinations, the 5 crane loading cases may be defined as a Group; then only one combination which includes 1.6*"Crane load group" need be defined. The program will then automatically generate a separate combination for each of the five loads in the group, i.e.
  - 1.4*Dead + 1.6*Imposed + 1.6*Crane (1)
  - 1.4*Dead + 1.6*Imposed + 1.6*Crane (2)
  - etc.

- Library:
  Note that standard combinations need not be redefined in every model as combinations containing groups may be stored in the library. For example:

  - 1.4*Dead + 1.6*Live
  - 1.2*Dead + 1.2*Live + 1.2*Wind
  - etc.

  For each Current Model, retrieve the standard combinations from the Library; then define groups that contain the "dead", "live" and "wind" load cases.

5.2 Element Coordinate Systems

The explanation of results for finite elements necessitates the introduction of two new coordinate systems in addition to the global and local element systems. They are:

5.2.1 Result Coordinate System X , Y , Z

The result coordinate system is the set of axes about which the element results are calculated and displayed. In most cases the results axes are identical to the local or global systems.

If the direction of the local x1-x2 axes for all of the elements are not co-directional, there will be an apparent inconsistency in the results and this will lead to time-consuming Mohr's circle calculations.

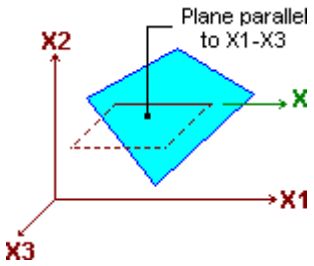
The program automatically assigns a default result coordinate system to each element. In general, the default X axis is as close as possible to the general direction of X1.

The default result X and Y axes are determined as follows:

- **Plane frame, plane grid:**
  The X axis is always parallel to the global X1 axis and the Y axis is always parallel to the global X2 axis.

  If the program discovers that the direction of the local x3 axis of an element is opposite to the direction of the global X3 axis, it reverses the sign of results for that element. This insures continuity of the lines.

- **Space frames:**
  - Elements parallel to the X1-X2 global plane:
    X = X1, Y = X2 , Z = X3
  - Elements parallel to the X1-X3 global plane:
    X = X1, Y = -X3 , Z = X2
  - Elements parallel to the X2-X3 global plane:
    X = X2, Y = X3 , Z = X1
  - Elements not parallel to a global plane:
    - X = direction parallel to the line of intersection of the element plane with a plane parallel to the X1-X3 global axis (+X in the general direction of +X1)
    - Y = perpendicular to X and in the general direction of X3. (+Y in the direction of +X2)



  - Elements perpendicular to the X1-X2 plane:
    - X = direction parallel to the line of intersection of the element plane with a plane parallel to the X1-X2 global axis.

In certain cases the default axes may not give the required results. For example:
- reinforcement in concrete slabs is not parallel to the global axes
- x3 local axis directions are not consistent in cylindrical or spherical models

You may revise the result coordinate system for each quadrilateral and triangular finite element:

Default

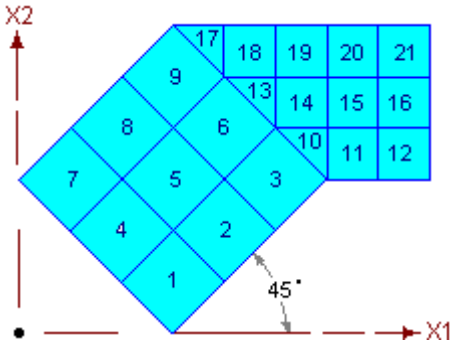
This option restores the default element result axes for the selected elements.

The element local axes

The result coordinate system of each selected element will be identical to its local coordinate system, i.e. X = x1 , Y = x2 , Z = +x3

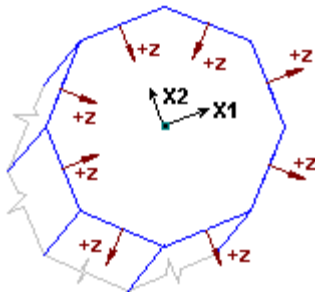
Examples:

- Grid:



For elements 1 to 9, the reinforcement is rotated 45° from X1 (i.e. parallel to the element boundaries) and so the moments should be relative to the local axes. By default the results will be relative to the global axes.

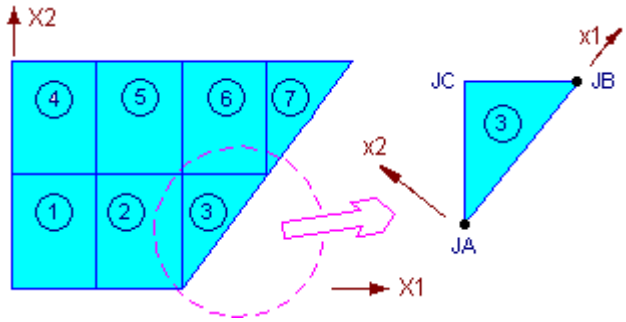
- Space frame:



The default Z axis directions are displayed above; note that some point inwards towards the centre of the cylinder while the rest point outwards. Use this option to define a consistent direction for all elements.

Note that this option may lead to misleading and inconsistent results. For example:

- Triangular elements:



The results for triangular elements 3 and 7 will not be relative to the same axes as quad elements 1,2,4,5,6.

- Slab reinforcement:
  Confusion will result in the reinforcement calculation if the local x3 axis directions are not consistent. The program assumes that the "TOP" face is the +Z face of the slab and the "BOTTOM" face is the -Z face. Hence, "Top" and "Bottom" reinforcement in elements may actually be at the same face if their x3 axis directions are reversed.

Local axes of a selected element

The result axes for an element may be defined as identical to the local axes of a different element. The "different" element must lie on a parallel plane.

For example, "local axes" were specified for a group of elements but their local x3 axes directions are not consistent. Confusion will result in the reinforcement calculation because the program assumes that the "TOP" face is the +Z face of the slab and the "BOTTOM" face is the -Z face. Hence, "Top" and "Bottom" reinforcement in different elements will actually be at the same face!

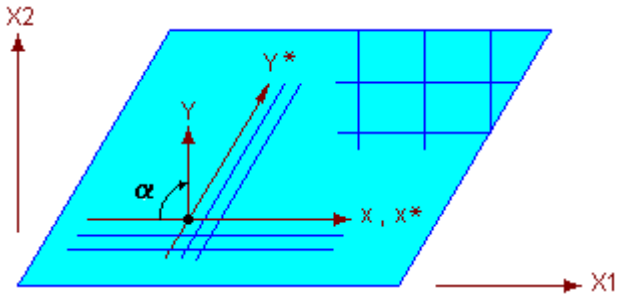
To unify the results, specify that the results axes for all elements are identical to the local axes of one element.

5.2.2 Reinforcement coordinate system

The reinforcement coordinate system is required for reinforcement design moment calculation in concrete slab models. The reinforcement axes X* and Y* are defined as parallel to the directions of the reinforcement. The program assumes that X* is always in the same direction as the result coordinate system X axis. Y* can be at any angle a from X*. Normally, $\alpha = 90°$, and the program assumes this value if no other value is defined.

The design moments Mx* and My* for reinforcement design are calculated according to the Wood and Armer equations in the direction of X*, Y*.

The following figure shows an example with $\alpha > 90°$.



The axes parallel to the main directions of reinforcement are denoted by X* and Y*. The program always assumes X* to be parallel to the result X axis.

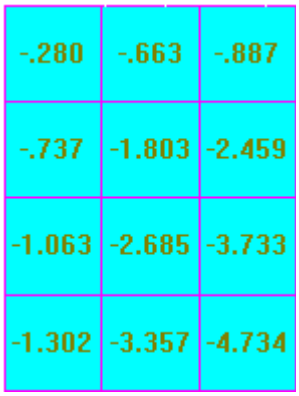
In most reinforced concrete slabs, the two reinforcement directions are perpendicular. This is the default assumption of the program.

For all other cases, the skew angle a between X* and Y* must be defined. The program will then calculate the design moments for reinforcement in the directions of the reinforcement coordinate axes X* and Y*.

5.3 Graphic Results

5.3.1 Results at Element Centres

To display the model geometry with the numerical value of the result written at the centre of each element. For example:



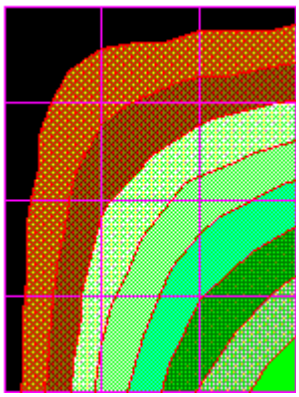
This option superimposes the tabular results at the element centre on the geometry display. The values are the same as those displayed in the tables.

Note:

- Results are displayed according to the Default element result coordinate system, unless the result system was revised for specific elements ("Options"/"Element results coord. system" in Menu bar)
- the program does not change the sign of the results, **even if the directions of the local axes are inconsistent**.
- local stress or moment concentrations at element corners, e.g. from a joint load applied at a node or a support reaction, are not displayed. Display Contour map or Results Along a Line for a more complete picture.

5.3.2 Result Contour Map

To display the model geometry with a contour map of the results superimposed. Each line of the contour map gives the location of a specified value of the result. For example:



This option creates a contour plot from the element centre and corner results. In order to produce smooth and continuous contours, the program averages the exact corner results from all the elements connected to a particular node, as well as along the edges of two adjacent elements.

Referring to the explanation for Results along a Line it is obvious that this averaging of results may lead to a discrepancy between the corner results from the tables and the contour map.

Results are displayed according to the Default element result coordinate system, unless the result system was revised for specific elements ("Options"/"Element results coord. system" in Menu bar)

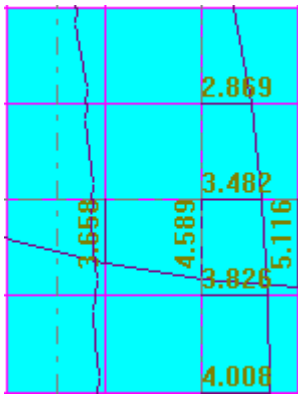
Space models:

***** WARNING *****

The local x3 axis direction is not reversed as in plane models. If the x3 axes of adjacent elements are in opposite directions, the sign of the results will be inconsistent and their interpolation over the element surface will be incorrect.

5.3.3 Results along a line

To display the results plotted along a section through the model. For example:



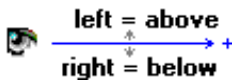
This option calculates the results at all points along any line arbitrarily drawn through the model, using the linear stress distribution assumed in each element. Therefore, this option will show a local stress or moment concentrations at element corners and the results correspond to those in the tables.

If a section is plotted along an element boundary, the program uses the results of one of the adjacent elements and does not average the values of all the adjacent elements.

The user selects the plot of the result type **along the line** or **perpendicular to the line**.

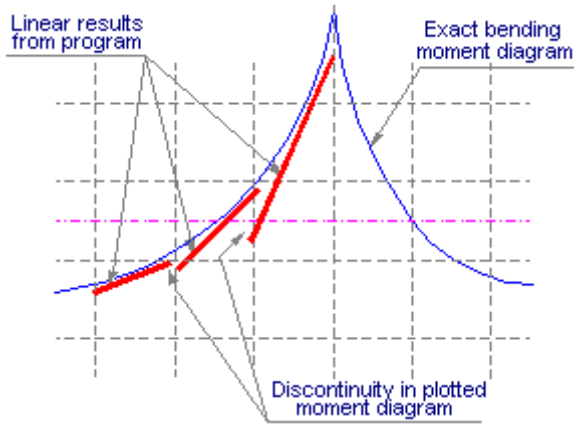
- **X** always refers to the axis along the line of the section.
- **Y** always refers to the axis perpendicular to X (in the element plane).
- **Z** axis is perpendicular to the plane of the element:
  - Plane models:
    The positive direction of the Z axis of all elements is the positive direction of the global X3 axis, i.e. if the local x3 axis of adjacent elements point in opposite directions, the result diagram will still be continuous.
  - Space models:
    The positive direction of the Z axis of each element is the positive direction of the local x3 axis of the element; if the local x3 axis of adjacent elements point in opposite directions, the result diagram will not be continuous.

The results are drawn 'above' and 'below' the line as follows:
- stresses: compression stresses are negative and are drawn below the line
- moments: moments are positive if they create compression stresses 'above' the element and are drawn below the line

The 'above' side is always to the left of the line when looking in its positive direction: 

The positive direction is determined as follows:
- line parallel to a global axis (within 5° of the axis):
  the positive direction of the axis
- plane model - line defined by 2 points:
  from the 1st point to the 2nd point
- space model - general case:
  in the general direction of +X1, except -
  - if perpendicular to X1: in the general direction of +X2
  - if perpendicular to X1-X2:in the general direction of +X3

Note that while the stress distribution within each element is assumed linear, the actual stress distribution is usually non-linear. This discrepancy results in discontinuities in the result diagram as shown below. Using a finer mesh in areas where the slope of the result diagram varies significantly will improve the accuracy of the results. An indication of the inaccuracy can be obtained by comparing results at the same node for adjacent elements - theoretically they should be identical but in general they differ.



5.3.4 Deflections

The program displays the numerical value of the deflection in the form *ddd* / 10^n, where *ddd* is written adjacent to the beam and the factor 10^n appears at the bottom of the screen.

Examples:

- **51** written adjacent to the element; **Values are * 10^3** at the bottom of the screen.
  deflection = 51/1000 = 0.051 (current length units).
- **272** written adjacent to the element; **Values are * 10^2** at the bottom of the screen.
  deflection = 272/100 = 2.72 (current length units).

5.4 Tabular results

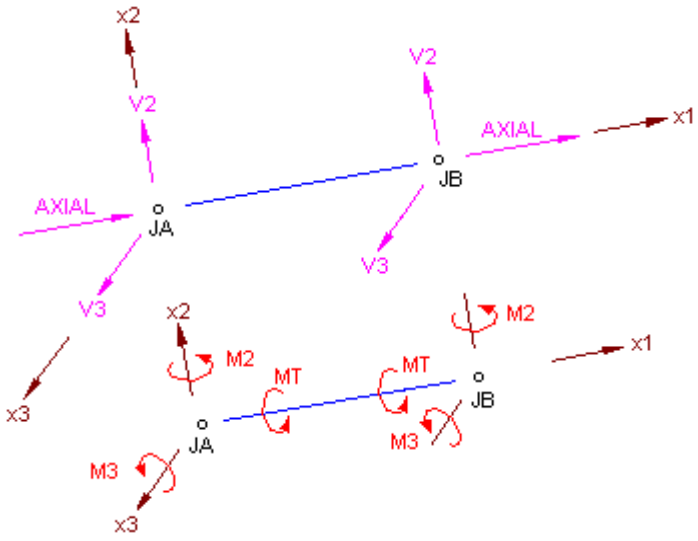
5.4.1 Tables - Sign Conventions

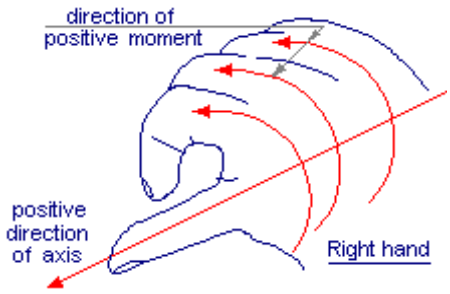
The tabular results use a mathematical sign convention as opposed to the standard engineering sign conventions; the sign of the results may often be opposite to what is expected and so the user must understand the conventions in order to correctly interpret the results.

5.4.1.1 Sign Conventions - Beams

Member results are listed at nodes JA, JB of each element. Results are relative to the local coordinate axes.

The positive sign conventions are:

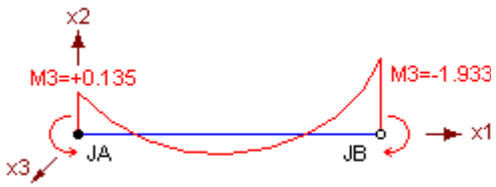




- **Moments:** (M2, M3, MT)

  The direction of a positive moment is determined by a right-hand rule. The thumb points in the positive direction of the local axis about which the moment acts, and the other fingers all curl in the direction of the positive moment.

  In the following example engineering negative moments act at both ends of the beam, however the sign of the moment at JA will be positive in the tables.



- **Shear:** (V2, V3)
  A positive shear force acts in the positive direction of the local axis.

- **Axial:**
  The sign of the axial force is positive in the +x1 direction of the beam. A positive axial force value at JA (along with a negative value at JB) always indicates that the beam is in compression. For trusses, only the result at JA is displayed, i.e. a positive value indicates compression! (the value at JB may be different if an intermediate axial load is applied, but the result will not be displayed).

5.4.1.2 Sign Conventions - Quadrilateral and Triangular Elements

The results are relative to the element result coordinate system. (Refer to 5.2)

The forces are per unit width. i.e, FX = 50.2 means 50.2 ton/m (kN/m, kip/ft, etc.).

- **STRESSES**:
  +SX = stress in element result X direction on the +Z surface.
  +SY = stress in element result Y direction on the +Z surface.
  +SXY = shear stress on +Z surface.

  -SX = stress in element result X direction on the -Z surface.
  -SY = stress in element result Y direction on the -Z surface.
  -SXY = shear stress on -Z surface.

- **FORCES**= stress resultants in the result coordinate system.

  The element forces are computed directly from the element stresses:

$$\begin{bmatrix} FX \\ FY \\ FXY \end{bmatrix} = \frac{T}{2} \times \left(\begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{+Z} + \begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{-Z} \right)$$

- **PRINCIPAL STRESSES** = principal stress at each face derived from the Mohr's circle equations:

$$MAX = \frac{SX+SY}{2} + \sqrt{\left(\frac{SX-SY}{2}\right)^2 + SXY^2}$$

$$MIN = \frac{SX+SY}{2} - \sqrt{\left(\frac{SX-SY}{2}\right)^2 + SXY^2}$$

  where: MAX and MIN are the algebraic maximum and minimum, not the absolute.

  S. MAX = $S.MAX = \frac{MAX - MIN}{2}$ = maximum shear

  ANGLE = the amount (in degrees) the element X axis must rotate counterclockwise about the Z axis to coincide with the principal stress axis.

$$= \frac{\pi}{2} \tan^{-1}\left[\frac{2 * SXY}{SX - SY}\right]$$

  Note:
  - when the X axis coincides with the maximum stress axis, Y coincides with the minimum stress axis.
  - MAX. SHEAR occurs on a plane offset 45° from the principal axis.
  - Shear is zero in the principal stress directions.

  The positive sign conventions for all stresses and forces are shown in the figures below.

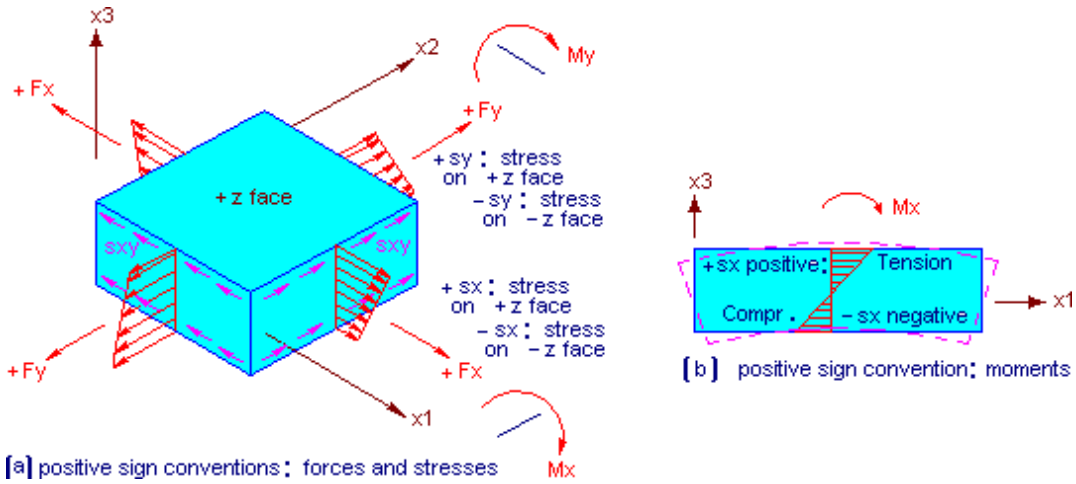
- **MOMENTS** = Moments relative to the result coordinate system at the centre of the element.

  The moments are computed directly from the stresses by:

$$\begin{bmatrix} MX \\ MY \\ MXY \end{bmatrix} = \frac{T^2}{12}^x \left(\begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{+Z} + \begin{bmatrix} SX \\ SY \\ SXY \end{bmatrix}_{-Z} \right)$$

  Note:
  - The sign convention for moments is illustrated in Figure (b) below: referring to the equation above for calculating MX, it is obvious that a stress distribution with tension on the +z face results in a **positive** bending moment.
  - MX is the moment in the direction of the element result X axis and not the moment about the X axis (see Figure below). Therefore, the stress SX corresponds to the moment MX. Similarly, MY is the moment in the direction of the element result Y axis and not the moment about the Y axis.
  - The moments are per unit width, i.e. MX = 50 indicates 50 t m/m (kN m/m, ft kip/ft, etc.)

The following are the positive element results sign conventions:



Referring to Figure (a) above, approximately equal and opposite forces act on opposite faces of the element. The sign of the displayed results are for the face in the positive direction of the result axis. Therefore:
- if FX is positive, the element is in tension in the direction of X.
- if +SX is positive, the top face is in tension.
- etc.

Note:
- for graphic results, the program may modify the sign of the results in order to ensure consistency, etc.

5.4.2 Beams - Axial force, Moments and Shears

Display axial force, shear force and moment values at both ends of beams and either the maximum value in the span or at specified intervals - $^1/_2$, $^1/_4$, $^1/_5$, $^1/_{10}$ or $^1/_{20}$ of the span. All results are relative to the beam local axes.

Example:

BEAM	COMB	NODE	AXIAL	V2	V3	MT	M2	M3
4	1	5	-0.076	1.550	0.000	0.000	0.000	0.135
		6	0.076	2.450	0.000	0.000	0.000	-1.933
	2	5	+0.089	0.765	0.000	0.000	0.000	0.325
		6	-0.089	-1.235	0.000	0.000	0.000	-1.263
		MAX	0.089	2.450	0.000	0.000	0.000	1.933
		COMB	2	1	0	0	0	1
		MIN	-0.076	-1.235	0.000	0.000	0.000	0.135
		COMB	1	2	0	0	0	1

where:

Axial = axial force along beam x1 axis. For trusses, only the result at JA is displayed (the value at JB may be different if an intermediate axial load is applied, but the result will not be displayed).
V2 = shear force parallel to the beam x2 axis
V3 = shear force parallel to the beam x3 axis
MT = torsion moment about the beam local x1 axis
M2= bending moment about the beam local x2 axis
M3= bending moment about the beam local x3 axis

Refer also to 5.4.1.1 - Sign conventions - beams.

If the **End results and max in span** option is selected, the display is:

2	5	+0.089	0.765	0.000	0.000	0.000	0.325
		FR=0.45	0.044				-2.325
	6	-0.089	-1.235	0.000	0.000	0.000	-1.263

where:

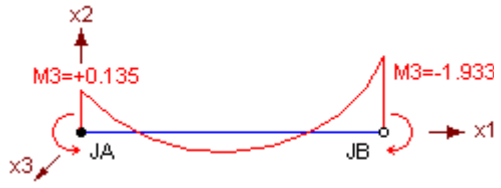
FR = fraction of span length (from JA) at which the maximum intermediate moment occurs.

Note:

- The intermediate moments are calculated at 1/20 intervals and at concentrated load locations.
- The sign of the intermediate moment and shear is relative to the sign at JA.
- The intermediate shear value displayed is at the same point as the maximum moment.
- An intermediate value is displayed only if the maximum positive moment or maximum negative moment are not at the end supports.
- All intermediate values displayed are included in the following MAX / MIN value searches.

When calculating the MAX/MIN results, the program reverses the sign of the moments at JB:

Referring to 5.4.1.1 - Sign Conventions - beams, and to the example in the following figure, it is apparent that engineering 'negative' moments at the two ends of the beam have opposite signs in the table.



To ensure consistency when calculating the 'maximum/minimum' results, the program reverses the sign of the moment at JB. Therefore, the moment results for the beam in the above Figure are displayed as:

BM	COMB	NODE	.....	.....	M3
22	1	43	.....		+10.00
			FR=0.45		-7.50
		44	.....	.....	-20.00
		MAX	.....	.....	+20.00
		COMB			1
		MIN	.....	.....	-7.50
		COMB			1

Note the sign reversal in the "MAX" table.

5.4.3 Span Deflections

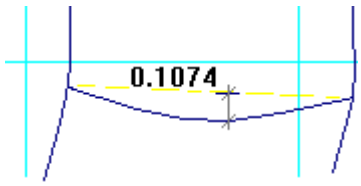
Display the maximum span deflection. The deflection is relative to the final location of the nodes, not the joint coordinates. For example:

BEAM SPAN DEFLECTION for combination 1

Exit Goto

Beam	x2	x3
5	-0.000057	-0.000001
6	-0.107460	-0.000001
9	-0.050046	0.000000
90	-0.000050	0.000000

For beam 6:



5.4.4 Element results

Element Moments, forces, stresses

- **Stresses**:
  ±SX = stress in element result X direction on the ±Z surface.
  ±SY = stress in element result Y direction on the ±Z surface.
  ±SXY = shear stress on ±Z surface.
- **Forces, moments** = stress resultants in the result coordinate system:

Note:
- the moments are per unit width, i.e. MX = 50 indicates 50 t m/m (kN m/m, ft kip/ft, etc.)
- MX is the moment in the direction of the element result X axis and not the moment about the X axis (see Figure below). Therefore, the stress SX corresponds to the moment MX. Similarly, MY is the moment in the direction of the element result Y axis and not the moment about the x2 axis.
- a stress distribution with tension on the +z face results in a **positive** bending moment; refer to 5.4.1.2 - Element sign conventions.
- For the equations relating moments and forces to stresses, refer to Element sign conventions.

Example (moments sorted by combinations):

El.	Comb	Mx	My	Mxy	Rot.
1	1	-.1030E+01	.5886E-05	.2443E+00	.0
	2	-.2060E+01	.1177E-04	.4886E+00	

- Maximum values are calculated from all of the result points.
- **Rot** indicates the angle between the local x1 axis of the element and the result X axis.

The result axes may be defined in the Element result coordinate system option. If no specific axes are defined, the program uses the default result axes. Refer to 5.2.1.

5.4.5 Reactions

The program sums at each node all of the applied loads and element result end forces of the attached elements for all degrees-of-freedom. The sum for degrees-of-freedom at nodes that were not defined as restraints should be equal to zero.

The reactions are:
X1, X2, X3 = reaction force parallel to the global axes. A positive force is in the positive direction of the axis.
X4, X5, X6 = moments about X1, X2, X3, respectively. For the positive direction of the moment, refer to 5.4.1- Sign conventions

The results displayed are in effect the "actions" on the support.

- **Results Sorted by Loads:**
  The program prints the sums only for the non-zero nodes. In effect, the sums are the reactions at the restrained nodes (note that they are actually the actions on the support). For example:

NODE	X1	X2	X3	X4	X5	X6
1	-2.591	-49.896	.000	.000	.000	.000
2	1.110	-58.305	.000	.000	.000	.000
3	.884	-31.546	.000	.000	.000	.000
4	-.093	-24.966	.000	.000	.000	.000
5	.117	-22.735	.000	.000	.000	.000
6	.572	-12.553	.000	.000	.000	.000
SUM	.000	-200.000	.000	.000	.000	.000

  The sum of the reactions displayed in the bottom line should equal the sum of the loads which were applied in that loading case.

  Due to the limited numerical precision of computers (and particularly micro-computers), non-zero values will occasionally appear in the Reactions for unrestrained degree-of-freedoms (i.e. nodes that were not defined as supports). Usually, these values will be negligible in comparison to the internal forces at these degree-of-freedoms (approximately N*10E-5, where N are the internal forces at a DOF). However in certain instances, the numbers could be much greater.

  In such cases, the user should check for the following causes:
  - Singularity messages during the solution stage.
  - Too large a difference between the values of the largest and smallest moment-of-inertia (or area) was defined for the model; The ratio between the largest and smallest values should not exceed 1:10E8.
  - In plate bending elements, the element thickness is large relative to the element dimensions (note that only the Reactions are affected - moments and deflections are accurate)

- **Results Sorted by Elements/Nodes**
  The reactions are printed only at nodes that were defined as restraints and so the inaccuracies described above will not be apparent.

Note that the max/min reactions are the maximum and minimum numerical reactions at the supports.

** Blank page **

6 Dynamic analysis

This dynamic analysis module analyses the modal shape of the model :
- solves for the natural frequencies and the corresponding mode shapes
- calculates the earthquake response and the resulting moments and forces in the model based on the calculated mode shapes and Code related factors
- calculates the time history response for forced vibrations.

6.1 Mode Shape and Natural Frequency Analysis

This dynamic analysis module analyses the modal shape of the model. The program solves for the natural frequencies and the corresponding mode shapes.

The program solves the problem of undamped free vibrations. This involves the solution of the generalized eigenvalue equation:

$$\mathbf{K}\cdot\boldsymbol{\phi}=\mathbf{M}\cdot\boldsymbol{\phi}\cdot\boldsymbol{\Omega}^2$$

where: $\mathbf{K}$ = stiffness matrix
$\mathbf{M}$ = Mass matrix
$\boldsymbol{\Omega}$ = eigenvalue matrix
$\boldsymbol{\phi}$ = corresponding eigenvector matrix

The eigenvalues correspond to the natural frequencies by the following equations:
eigenvalue = w^2
natural frequency = $w/2\pi$

Each value of the eigenvector is the relative displacement of the corresponding degree-of-freedom.

The mode shape of the lowest frequency (longest period) will be numbered 1 and all the others will be numbered respectively in ascending order.

The program assumes a lumped-mass model, i.e. the mass of the model is concentrated at the nodes.

The program solves the general eigenvalue problem using the Subspace Iteration method. Explained simply, the program bases the calculation of the eigenvalues in the current iteration on the eigenvalues of the previous iteration. When the difference between the previous and current values is very small, the program terminates the iteration process.

6.2 Seismic Analysis

This module calculates the earthquake response and the resulting moments and forces in the model based on the calculated mode shapes and Code related factors.

The mode shape analysis calculates 'n' different mode shapes. The maximum response (deflection, base shear, etc.) for each shape is calculated from a "Response Spectrum". This spectrum is a graph which gives the acceleration as a function of the natural period, T, of the model.

The spectrum may be an idealized one taken from a Code, e.g. Figure 1B in the SEAOC (California Blue Book) Code, or it may be based on ground motion histories at the specific site.

When calculating the maximum response, the total response usually cannot be obtained simply by adding the maximum responses of the individual nodes because these maxima usually do not occur at the same time.

The user may select one of the following methods to estimate the maximum total response from the maximum calculated modal values;

- SRSS (Sum of Root of the Sum of the Squares) method.
- CQC (Complete Quadratic Combination)

In both methods the program calculates the response for each mode separately and then combines them according to a formula that accounts for the fact that when one mode achieves its maximum response, the responses of the other modes are less than their individual maxima.

Note that each maximum response is calculated separately, e.g. the maximum moments are calculated as the RSS/CQC of the moments from the individual mode shapes, and not as the moments resulting from the RSS/CQC deflections of the model.

- SRSS (square root of sum of squares)
  The estimated response R (force, displacement, etc.) at a specified coordinate may be expressed as:

  $$R = \sqrt{\sum_{i=1}^{N} R_i^2}$$

  where Ri is the corresponding maximum response of the *i*th mode at the coordinate.

- CQC (complete quadratic combination)
  The estimated response may be expressed as:

  $$R = \sqrt{\sum_{i=1}^{N}\sum_{j=1}^{N} R_i^2 \rho_{i,j} R_j^2}$$

  where the cross-modal damping coefficient ρ may be approximated by:

  $$\rho_{i,j} = \frac{8\xi^2(1+r)\, r^{3/2}}{(1-r^2)^2 + 4\xi^2 r(1+r)^2}$$

where: $r = \omega_j/\omega_i$ = ratio of the natural frequencies of modes i and j
ξ =the constant modal damping

Application of the SRSS method generally provides an acceptable estimation of the total maximum response. However when some of the modes are closely spaced, the method may grossly underestimate or overestimate the maximum response. Large errors have been found in particular in space models in which the torsional effects are significant. The term "closely spaced" may be arbitrarily defined as the case where the difference between two natural frequencies is less than 10% of the smaller frequency.

The CQC method is a more precise method of combining the maximum values of modal response.

Note that the two methods are identical for undamped models (ξ= 0).

The program also calculates:
- Mass center and rigidity centre of each level
- Maximum/minimum deflections at each level and the story drift

For additional theoretical explanations and background, refer to any textbook covering dynamic response of multi degree-of-freedom systems.

6.3 Forced Vibrations and Time History Response

This module calculates the transient (history) response of a model subject to dynamic loads in which viscous damping is present. It enables the dynamic analysis of models subject to impact, impulse or cyclic loads or any other type of load that varies with time.

The equations of motion are solved on the basis of the results from the Natural Frequency and Mode Shape analysis.

The equations of motion are:

$$[m]\{\ddot{x}\} + [c]\{\dot{x}\} + [k]\{x\} = \{P\}\,F(t)$$

where:

[m] = diagonal mass matrix
[c] = matrix of damping coefficients
[k] = stiffness matrix
{P} = joint load distribution
F(t) = time history of the applied forces

The program assumes:

- At each node the history behaviour of the load is represented by the input joint loads multiplied by the history function F(t).
- the history function F(t) is composed of either:
  - a series of straight line segments defined by a set of pairs of time and amplitude values:
    {t1, F(t1)}, {t2, F(t2)}, ... , {tn, F(tn)}
    where: tn > tn-1 > 0
  - a sine function curve

  A different history function may be defined for each load case.
- the damping matrix [c] is proportional to the mass matrix:
  [c] = 2 [?] [m]
  where [?] is a constant diagonal matrix.
  The damping is defined as a set of coefficients, one for each mode shape, where the coefficients represent a percentage of the critical damping ?cr = wn (wn = modal frequency).

** Blank page **

7 Structural Steel Design

7.1 Steel Design - General

The ***STRAP*** Steel design module is a program for the design of structural steel buildings.

The program selects for each member the lightest section which meets the Code requirements. The sections are chosen from a specified table. Section constraints, design data and intermediate supports may be defined.

Alternatively, the program may be instructed to check the suitability of a section specified by the user.

The program automatically searches for the critical load combination, and checks the section for bending moments, lateral-torsional buckling, shear, axial forces and combined stresses as well as deflections and slenderness.

The program may be instructed to select sections according to the methods outlined in any one of the following structural steel design codes. For more information on the Design Assumption, refer to:

- British Standard BS 5950- Part 1 : 2000, "Structural Use of Steelwork in Buildings".
  Part 3, Section 3.1 - "Design of Simple and Continuous Composite Beams"
- Eurocode: Eurocode 3 - Design of Steel Structures - Part 1 , 1992.
  Eurocode 4 - Design of Composite Steel and Concrete Structures - Part 1, 1990
- AISC - Load and Resistance Factor Design - 1993.
- AISC - Allowable Stress Design - 1989.
- CSA/CAN S16.1 - Limit States Design of Steel Structures - 1994.
- AASHTO - LRFD Bridge Design Specifications - 1999.
- AASHTO - Standard Specifications for Highway Bridges (ASD) - 1996.
- SABS 0162-1: 1993.
- IS:800 - Code of Practice for General Construction in Steel:
  - 1984
  - 2003 (LRFD)
- GBJ 17 88 - Specification for the Design of Steel Structures - 1988.

Cold formed:

- AISI - Specification for the Design of Cold-Formed Steel Structural Members.
  - 1986 Edition with 1989 Addendum - ASD only
  - 1999 Edition - ASD and LRFD
- CSA S136 - 94.
- Eurocode 3 - Part 1.3., 1996.

The program strives to design the lightest structure possible; the section chosen is the one with the least self-weight that satisfies all of the design criteria for all loading combinations and meets the Code requirements.

For all Limit States Codes (all Codes except AISC/AASHTO/AISI - ASD and IS:800):

The factored combinations may be defined in *STRAP* loading or in the "Combinations" option after the solution. The choice does not affect any of the design calculations except for deflections which are based on service loads. The module assumes that CASES are unfactored, i.e the combinations were defined after the solution. If the factored combinations were defined in Loading, the deflection results for the load cases will also be factored. The allowable deflection limit should then be increased proportionally.

7.2 Steel section tables and selection

The program contains several complete section tables - the "master" table.

- British sections (CONSTRADO tables)
- American sections (ASTM)
- European sections (Euronorm)
- Canadian
- South African
- Indian

Cold-formed sections may be added to the master tables.

In addition:
- A "user" steel table may be created. This table can contain sections from any or all of the three tables above, or user-defined rolled/cold-formed sections. Refer to 1.5.2 - User steel table for instructions.
- additional “built-up" sections may be defined by specifying dimensions; the program will assume that these sections are welded shapes.
- Combined (rolled) sections may be defined (2L, I+[,][, etc).

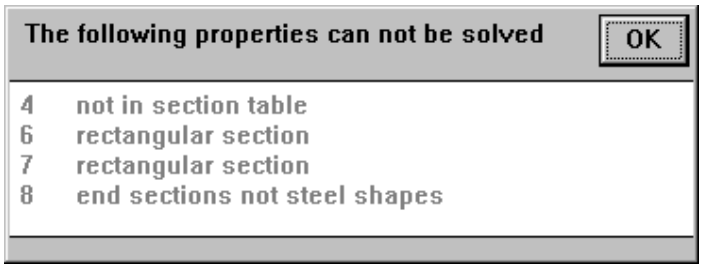
The following terminology is used throughout the program:
- Master table: The initial complete table of steel sections (British, American, European, etc. **or** User).
- Model table: The section table for the current model, containing sections from the selected master table.
- Section type: Section classification according to shape, e.g. CHANNEL, RHS, Z+LIPS, etc.
- Group: A user-defined list of sections which may contain sections from several types.

The program either selects a suitable section from a list or checks a specified section.

In the following cases, the program will by default check the section defined in ***STRAP*** geometry:
- if the section properties of the member were defined as a steel section using the **Steel table** option.
- “combined” sections, defined in the ***STRAP*** geometry or in the Steel Postprocessor
- if the section properties of the member were defined using the **Define the section dimensions** option, the program will assume that the member has a Built-up welded section with the same dimensions.
  Note that only "Pipe", "Tubes", "L shape" and symmetric "[shape", "I shape" and "T shape" sections will be converted to Built-Up sections.
- if a "tapered section" was defined in ***STRAP*** and the properties at both ends of the member were defined using the **Define the section dimensions** option, the program will check the tapered section.

For all other ***STRAP*** properties, the program will ignore the ***STRAP*** properties and must be instructed to select the lightest suitable section from a user-defined list or to check a specified section. The cases are:
- "dimension" section types - "Rectangular" and unsymmetric "[shape", "I shape" and "T shape" sections.
- properties defined with the section constants "**A=,I=**" option
- tapered sections defined with properties other than acceptable "built-up" shapes.
- if the property was defined with a steel section not in the current model table.

A warning will be displayed at the start of the program. For example:


The following properties can not be solved
OK
4 not in section table
6 rectangular section
7 rectangular section
8 end sections not steel shapes

When selecting a suitable section, the program checks sections from a specified list, beginning from the lightest one, until it finds the first adequate section. If the program had to search the complete "master" table when designing each member in the model, the selection process would be very time consuming.

7.3 Creating a Steel Structure from the *STRAP* Model

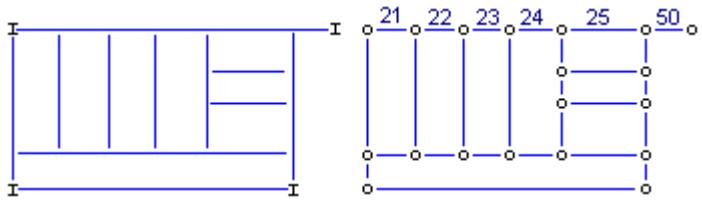
Member selection is automatic; the program designs all beams in sequence without prompting for information.

Therefore, all data and information relevant to the design of the members as required by the Code must be entered before member selection begins.

In many cases the model geometry as defined in ***STRAP*** does not provide sufficient information for the Code to carry out an accurate design.

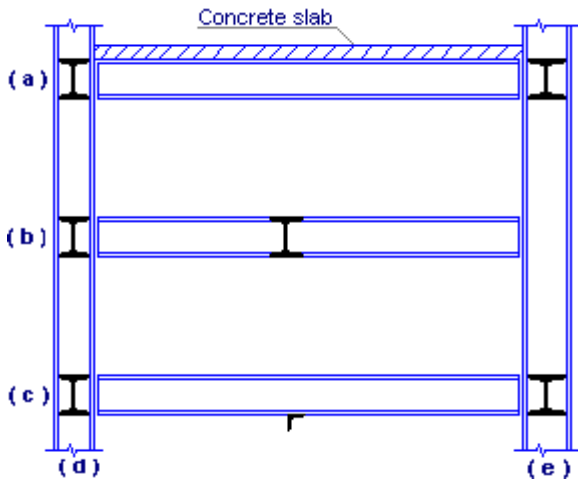
For example, Figure (a) shows a typical steel floor plan. Figure (b) shows the same floor as analyzed in ***STRAP***. It is obvious that the program is unable to determine which ***STRAP*** members form continuous beams, i.e. which chains of members must be designed as a single unit by the program.

For example, members 21-22-23-24-25-50 form a single beam.



(a) (b)

Another important item that must be defined in the module is the location of "intermediate supports". The support locations are required by the program in order to automatically determine the unsupported length for beams and the design length for columns.



- beam (a) has a continuous support on its upper flange, and is unsupported along the entire bottom flange.
- beam (b) has a single support on its upper and lower flanges.
- beam (c) is unsupported on its upper flange and has a single support on its lower flange.

If the upper flange is entirely in compression, the section required for each of the beams (a),(b),(c) will be different. As the support data is not available from the ***STRAP*** geometry, it must be defined in this module.

Similarly, column (e) has an unsupported length about its minor axis double that of column (d).

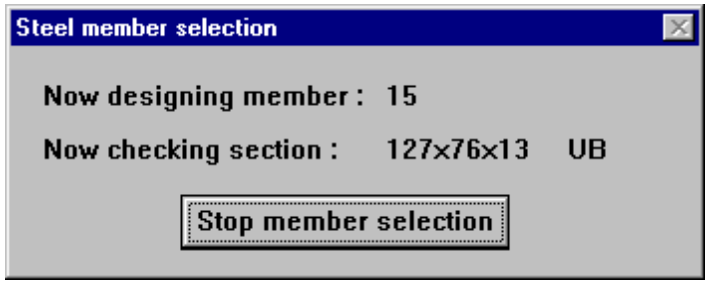
Other data items required for design are:
- allowable maximum deflection
- allowable maximum slenderness (tension and compression)
- beam end support conditions
- column effective length factors
- section orientation
- steel grade

Additional design constraints for section selection may also be specified:
- minimum and/or maximum section dimensions for each member.
- a series of members may be defined as "identical", i.e. the same section will be selected for all members in the series.

7.4 Compute

When the option "Compute" is selected in the Main Menu, the program begins automatic member selection.

The progress of the design is displayed on the screen:



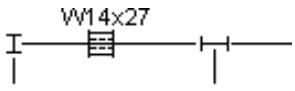
The program begins the design by checking the lightest section available. If the check fails (because of inadequate capacity or failure to meet user-defined constraints), the program selects the next heaviest member in the list and begins the check again. The process is continued until an adequate section is found.

Note:

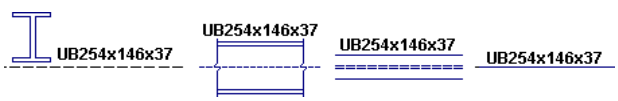
- if the type assigned to a member in the Section option is a cold-formed section, the program automatically designs the member according to the specified cold-formed Code.
- If more than one type is allowed (i.e. if there is more than one type in a group), the program searches for an adequate section independently for all of the types and then selects the overall lightest adequate section.
- if a member is part of an identical list, the program begins the design check from the section selected for the previous member in the list. If a larger section is required for the current member, the program will recheck all of the previous members in the list for the new section.
- refer to the extended manual for detailed explanations on the Code equations used by the program.
- refer to the extended manual for a detailed explanation on the method used by the program to select steel joists (American steel table only).
- For combined forces, the program may be instructed to either calculate the combined forces at 11 points along the length of the beam, using the actual forces at each point, and then use the worst case for design, or to combine the maximum result from each type, even though they may not be at the same location.
- two options are available for the design of angles and other doubly unsymmetric sections (created in the Section editor utility): the program may use either the principal axis properties Iu, Iv or the major/minor axis properties Ix,Iy.
- For general non-symmetric sections (e.g. T-sections), the program by default combines the maximum stresses even if they are not at the same location on the section, unless the exact flange location is specified.

7.5 General arrangement drawing

Create a "General arrangement" drawing (Line diagram) for any plane in the model. For example:



The section shape and name may be imposed on the beam line using one of the following four methods:



Note:

- the drawing may be generated on any plane, e.g. plans or elevations. If more than one plane is displayed on the screen when this option is selected, the program will prompt the user to select a plane defined by three nodes.
- the program writes the section name adjacent to each member. The name is written only once for a string of identical sections. The text size is specified by the user.
- The program differentiates between primary and secondary beams and terminates the line of the secondary beams before the intersection with the primary beams

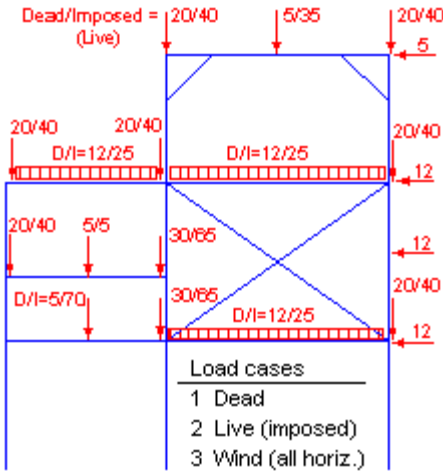
** Blank page **

7.6 Example

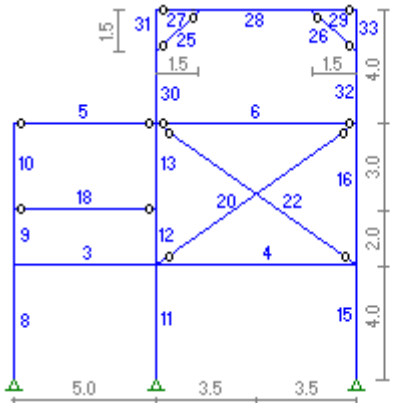
Design the plane frame shown below.

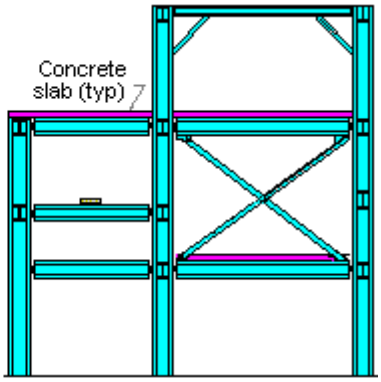
Note:

- This example illustrates the method of application of several steel design options. The example is intended as an aid to learning the proper use of the program options, and is not intended as a guide to proper engineering judgment in the construction of a model for design.

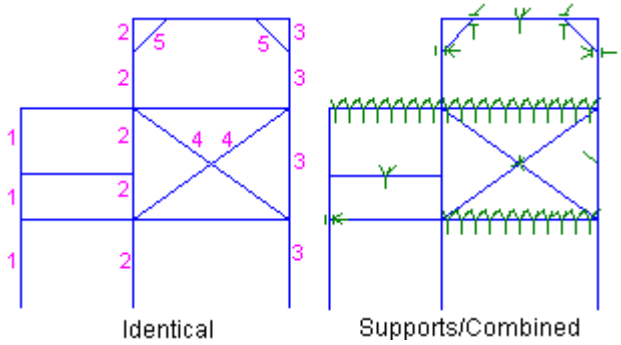


Load combinations	for AISC-ASD
1=1.4D+1.6L	1=1.0D+1.0L
2=1.2D+1.2L+1.2W	2=0.75(D+L+W)
3=1.2D+1.2L-1.2W	3=0.75(D+L-W)
4=1.0D+1.4W	4=0.75(D+W)
5=1.0D-1.4W	5=0.75(D-W)





After the geometry and loads were defined in *STRAP* and the model was solved, the following data was defined using the Steel design module options:



"**Section**" :

Limit selection to:

Code	BS5950	Eurocode 3	AISC
Columns	UC	HEA	W
Beams	UB	IPE	W
Diag. Bracing	EQ.D.ANGLE	L	2L
Haunch beams	CHANNEL	UNP	C

Same section:

- Column 8-9-10 specified as an identical group Column 11-12-13-30-31 specified as an identical group
- Column 15-16-32-33 specified as an identical group
- Bracing 20 and 22 specified as an identical group
- Haunch beams 25 and 26 specified as an identical group

Supports:

- Members 4,5,6:
  "Continuous" support for +z major axis bending, major and minor axis buckling. The support is provided by the floor slab. The continuous buckling support for both axes cancels the axial force check.
- Member 16:
  Minor axis buckling support at mid-span. The support is provided by the beam perpendicular to the plane of the frame.
- Member 18:
  Support at midspan for +z major axis bending, major and minor axis buckling. The support is provided by the walkway passing over the beam. This support may be defined using the option "Define supports at concentrated load points".
- Members 20,22:
  Support at the midspan of the bracing for major and minor axis buckling (assumes that the bracings are attached).
- Member 28:
  Support at midspan for +z major axis bending, major and minor axis buckling. This support may be defined using the option "Define supports at concentrated load points".

Combined beams:

- Members 8,9:
  Beam 3 provides buckling support only for the major axis of this column, i.e. members 8 and 9 act as a single unit for minor axis buckling. Define major axis buckling and -z bending supports at the common node.
- Members 27,28,29:
  These three members form a single beam. Define major axis buckling and -z bending supports at the common node.
- Members 30,31:
  Haunch beam 25 provides buckling support only for the major axis of this column, i.e. members 30 and 31 act as a single unit for minor axis buckling. Define major axis buckling and -z bending supports at the common node.
- Members 32,33: Similar to 30, 31. Define major axis buckling and +z bending supports at the common node.

8 Concrete Design

8.1 General

The *STRAP* Concrete Postprocessor is a program for the design of reinforced concrete buildings.

The program designs beams, columns, walls and slabs according to the methods outlined in any one of the following reinforced concrete design codes . For more information on the Design Assumptions and Method of Calculation, refer to:

- British Standard BS 8110 - Part 1 : 1985, "Structural Use of Concrete"
- Eurocode 2 (EC2) - "Design of Concrete Structures" - Part 1 , 1991
  Eurocode 8 (EC8) - "Design Provisions for Earthquake Resistance of Structures" - Part 1-2, 1995
- ACI 318-05 - "Building Code Requirements for Reinforced Concrete" - refer to Appendix A8.3
- CSA A23.3-94 - "Design of Concrete Structures"
- IS:456 – 2000 – "Code of Practice for Plain and Reinforced Concrete"
  IS:13920 - 1993 - "Ductile Detailing of Reinforced Concrete Structures Subjected to Seismic Forces"
- NBr 6118-2000 - Brazil.

Note that the manual uses the following terminology:

MEMBER - refers to a *STRAP* beam element.

BEAM - refers to a continuous beam consisting of a series of connected **MEMBERS** defined in this postprocessor. "Beams" are defined by the user.

COLUMN - refers to a column consisting of a series of connected **MEMBERS** defined in this postprocessor. "Columns" must be defined by the user.

SPAN - refers to a span (between supports) in a **BEAM** or **COLUMN,** that may be comprised of more than one **MEMBER.**

WALL - refers only to a wall defined using the *STRAP* "Wall" option; Walls are identified automatically by the program. Quad and triangular elements cannot be designed as walls.

SLAB - refers to a plane of quad and/or triangular elements; Slabs are identified automatically by the program.

The terms **STIRRUPS** (American) and **LINKS** (British) are identical.

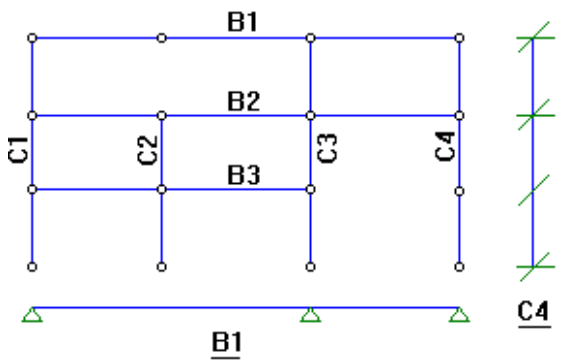
8.1.1 Creating a concrete structure from a STRAP model

The Concrete Postprocessor must design beams, columns and walls separately because the design methods and reinforcement calculation methods are totally different.

- All continuous beams and columns may be manually identified by the user or automatically by the program; the program uses the information from *STRAP* geometry for determining support locations and widths and section dimensions, if possible. The program then designs the beams and columns according to user specified parameters.
- Walls are identified automatically by the program. Quad and triangular elements cannot be designed as walls.

The following figure shows a simple plane frame analysed in *STRAP*. In order for the Concrete Postprocessor to design the structure, beams B1 - B3 and columns C1 - C4 must be identified so that the program can apply all of the relevant Code clauses for continuous elements.

For example, Beam B1 and Column C4 must be defined ignoring the dummy nodes.



8.2 Sections

The program converts the *STRAP* sections into design section as follows:

- Steel sections, sections defined by properties, tapered sections, tubes and pipes are considered as "Undefined"; **the program will not design beams or columns with undefined properties in any of the component spans.**

Beams:

- Rectangular and L-sections: dimensions are maintained.
- T and I-sections:
  - major axis - converted as defined
  - minor axis - converted to rectangular section with similar dimensions
- U sections:
  - major axis - converted to rectangular section with similar dimensions
  - minor axis - converted to I-section
- If the *STRAP* section is not symmetric (T, U and I-sections), the postprocessor creates a symmetric section using the minimum dimensions.

Columns:

- Rectangular, round, and L-sections: dimensions are maintained.
- T and U-sections; If the *STRAP* section is not symmetric, the postprocessor creates a symmetric section using the minimum dimensions.
- I-sections are considered as "Undefined".
- Solid sections created in *CROSEC* (section generator) are accepted and the dimensions are maintained.

8.3 Seismic design - general

This section explains the general principles of seismic design for reinforced concrete frames common to all design Codes.

In general, seismic design must insure minimum levels of ductility in the beams and columns and so has more stringent requirements for minimum reinforcement. In addition, much of the design is based on the moment capacity of the members rather than the design forces calculated by the analysis. The moment capacity must be calculated from the actual reinforcement in the beams/columns and so the program allows the user to increase the theoretical areas to the actual areas as detailed.

The design procedure is summarized as follows:

- beam longitudinal reinforcement is calculated from the design forces, but not less than the minimum specified by the Code
- beam shear reinforcement is calculated from shear forces derived from the moment capacity of the beam (but not less than the design forces).
- column longitudinal reinforcement is calculated from the design forces, but may be increased to ensure that the sum of the column moment capacities at a joint exceeds the sum of the beam moment capacities.
- column shear reinforcement is calculated from shear forces derived from the moment capacity of the beams connected to the ends of the columns (but not less than the design forces).

This method ensures a hierarchy of strengths of the different members. Note that three different moment capacities are calculated by the program:

- factored: normal capacity for non-seismic members
- nominal: capacity calculated using concrete and steel strengths not reduced by Code factors
- probable: capacity calculated using increased steel strength, i.e. actual conditions

and each calculation uses the appropriate capacity.

The design of both longitudinal and transverse reinforcement in columns is dependent on the capacity of the beams. Therefore, it is mandatory to compute the beams prior to computing the columns.

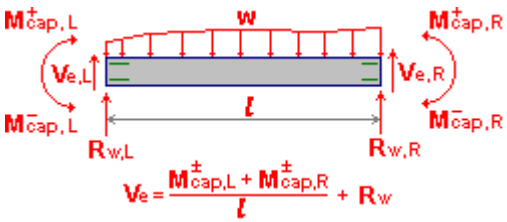
8.3.1 Beams

Moment:
The beams are designed for all load combinations as defined by the user.
In addition, the program complies with the following requirements found in all Codes:

- at support, the positive moment capacity is not less than a specified percentage of the negative moment capacity
- at any point along the beam, the positive and negative moment capacity is not less than a specified percentage of the negative moment capacity at the support

The program adds additional top/bottom reinforcement as required.

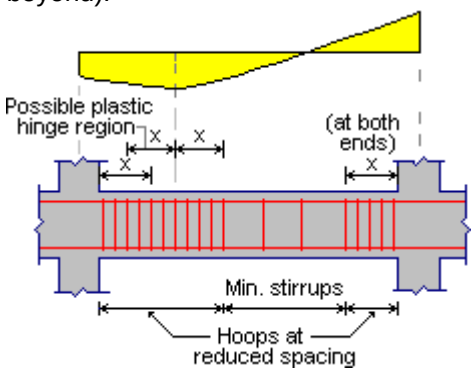
Shear:
The seismic design shear forces (Ve) are calculated from the probable moment strength of the beam together with the factored beam loads:



The program also checks the beam for the design shear from all load combinations.

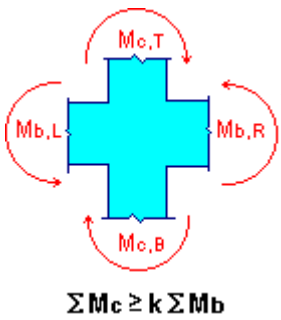
Note that **Mcap** is calculated from the actual reinforcement; it is important that the user increase the theoretical values to reflect the actual detailing.

Stirrups (links) are calculated from **Ve** , subject to the minimum requirements in the Code. In general, closed hoops at reduced spacing are required at all locations where plastic hinges may form (and to a distance x' beyond):



8.3.2 Columns

Moment and axial load:
The columns are designed for all load combinations defined by the user. Column areas are not 'reduced' for lightly loaded columns.



In addition, all codes require that the sum of the column flexural strengths at a column-beam joint exceed the sum of the nominal beam flexural strengths (strong column - weak beam), i.e.

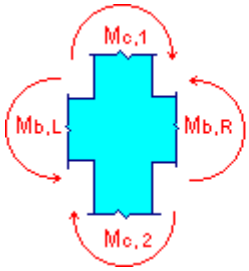
where:

- **Mb** is calculated from the actual beam reinforcement at the support (may be increased by the user)
- **k** is specified by the Code.

The program ensures that the columns comply with this requirement using the following procedure:
- Σ**Mb** is calculated at the joint:
- the moment is then apportioned to the columns above and below according to their relative stiffness
- the resulting moment is then applied as a separate load 'case' at the column top/bottom along with the factored axial load; additional reinforcement will automatically be added to the column if this requirement governs.

These load cases are marked as 'seismic' in the column extended detailed results.

Shear:
Similar to beams, the seismic design shear forces (V_e) are calculated from the probable moment strengths of the beams framing into the columns. The sum of the beam moment capacities at a joint are apportioned to the columns above and below according to their stiffnesses:



$$M_{c,1} = \Sigma M_b \times \frac{\text{column stiffness,1}}{\Sigma\,\text{column stiffnesses}}$$

and **V_e = (Mc,T + Mc,B)/L**

Stirrups (links) are calculated from **V_e** , subject to the minimum Code requirements. In general, closed hoops at reduced spacing are required at all locations where plastic hinges may form (and to a distance x' beyond).

At the base of the column, the program calculates **Mc** based on the column capacity because no beams frame into the node:

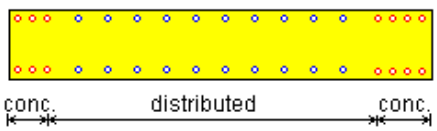
$M_c = M_d$ * capacity factor * f_y/f_{ye}

where: **M_c * capacity factor** = approximate column capacity based on design loads
f_y = steel strength reduction factor for normal design calculations
f_{ye} = steel strength factor for seismic capacity calculations

For example, a column at the base was designed for a moment = 46.2 kn-m and the capacity factor for the actual reinforcement = 1.07. Steel strength used for regular design = 0.87fy while 1.25fy is used for seismic capacity probable strength: **M_c = 46.2 * 1.07 * (1.25/0.87) = 71.0 kn-m**

8.3.3 Walls

The program calculates the distributed reinforcement on both wall faces and in addition determines the concentrated reinforcement required at both wall ends according to the requirements of the relevant seismic Code. The area where the concentrated reinforcement is required is also calculated.



8.4 Design Procedure

8.4.1 Non-seismic

For beams, columns and walls not designed for seismic forces:

- Define all continuous beams/columns including support locations and support widths. Walls are identified automatically be the program *(STRAP* wall elements)
- Define all properties not specified in *STRAP* geometry by dimensions.
  Beams: The program designs rectangular, tee, inverted tee, L, inverted L, and I shaped sections.
  Columns: The program designs rectangular, L-shaped, round, symmetric U and T shaped sections and any 'solid section' created in the *CROSEC* section generator..
  Note that other shapes are converted automatically by the program or ignored; refer to Define.
- Specify the default parameters for all beams/columns/walls.
- Define parameters for specific beams/columns, if different than the default parameters.
  Examples of parameters that may be defined:
  Beams: reinforcement and concrete type
  stirrup parameters
  moment redistribution percentage
  shear reduction, etc.
  Columns: reinforcement and concrete type
  Walls: effective length factors
  allowable bar diameters
  structure type: braced/unbraced, etc.
- Compute the results:
  The beams/columns/walls are designed in sequence without any prompts for information by the program. Therefore all design data and parameters must be defined before the design begins.

  The program carries out the following calculations:

  Beams:
  - calculation of moment and shear envelopes from all load combinations.
  - automatic moment redistribution (optional).
  - shear reduction at span supports (optional).
  - calculation of reinforcement steel required at all supports and spans.
  - automatic stirrup detailing with variable spacing.
  - deflection check - span/depth ratio (BS8110, EC2, IS456, NBr)
    deflection based on effective moment-of-inertia (ACI, CSA)

  Columns:
  - determination of the critical *STRAP* load combination
  - calculation of magnified (additional) moments for slender columns/walls.
  - selection of reinforcement arrangement that is able to withstand applied bending moments and axial force for all load combinations.

  Walls:
  - The capacity is calculated separately for *each* segment in the wall -
  - determination of the critical *STRAP* load combination
  - calculation of magnified (additional) moments for about the weak axis and minimum moments .
  - selection of reinforcement arrangement which is able to withstand applied bending moments and axial force for all load combinations, subject to minimum Code requirements. The reinforcement is distributed equally on both wall faces.

- Revise parameters, properties, etc. and compute again.
- Create column drawings and tables
- Print results.

8.4.2 Design procedure - seismic

The design procedure is more rigorous for models designed for seismic loads:

General:
- Define all continuous beams/columns including support locations and support widths.
- Define all properties not specified in *STRAP* geometry by dimensions.
- Specify the default parameters for all beams/columns. In particular, specify the seismic frame type and identify the seismic load cases (click the Seismic tab).
- Define parameters for specific beams/columns, if different than the default parameters.

Seismic:
The Codes specify the following hierarchy for the calculation:
- beam longitudinal reinforcement is calculated from the design forces, but not less than the minimum specified by the Code
- beam shear reinforcement is calculated from shear forces derived from the moment capacity of the beam, based on the actual reinforcement area (may be modified by the user), but not less than the design forces.
- column longitudinal reinforcement is calculated from the design forces, but may be increased to ensure that the sum of the column moment capacities at a joint exceeds the sum of the beam moment capacities ($\Sigma Mc/\Sigma Mb > k$).
- column shear reinforcement is calculated from shear forces derived from the moment capacity of the beams connected to the ends of the columns (but not less than the design forces).

Note that beams must be computed prior to columns as the column capacity is dependent on the end moment capacities of the connecting beams.

The design procedure is as follows:

Beams:
- Compute the beams - select

- Check results, revise parameters, properties, etc, if necessary, and compute again.
- Specify the exact (increased) reinforcement at all beam ends, top and bottom; select

or

and click the Modify Reinf. tab. To check, click **Data tables** and select **Display reinforcement table**.
- Display the reinforcement areas and the corresponding moment and shear capacities:
- click **Data tables** and select **Display seismic capacity table**.

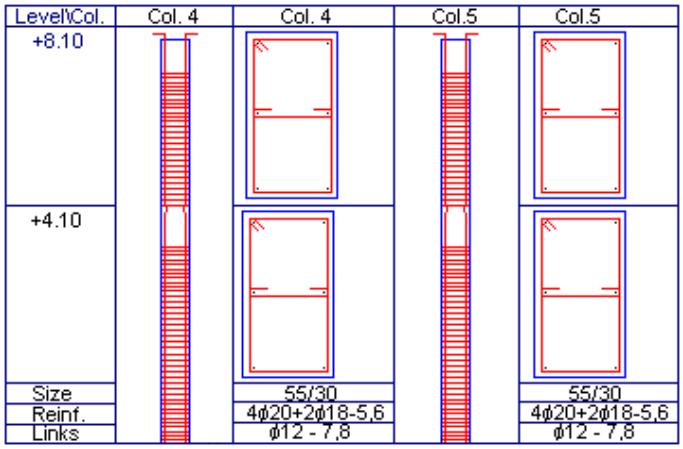
Columns:
- Compute columns - select

- To display data used to calculate links/stirrups, click **Data tables** and select **Display column shear table**.
- Check results, revise parameters, properties, etc, if necessary, and compute again.

8.5 Draw columns

Detail the reinforcement in computed columns and create drawings showing elevations and sections:

- lap lengths, no. of links, etc., are determined according to the relevant Code clauses
- the detailing may be drafted according to either of two options:
  - "drawing"- the sections and elevations are arranged on an engineering drawing
  - "table" - the sections and elevations are arranged in tabular form. For example:



Note that the default parameters for the drawing/table are assigned to the column when the columns are created; the parameters can be subsequently revised only by using the "Drawing parameters" option in the side menu. Creating the column again will restore the default parameters.

Bar schedule:

- Drawings:
  The schedule is added to the drawing and includes all the reinforcement currently displayed.
- Tables:
  The schedule is created in a separate file and may be viewed, edited and printed using the BARSW program.

8.6 Draw slabs

The program calculates and details the slab reinforcement, either as an arrangement of individual bars, a pattern of prefabricated meshes, or a combination of both.

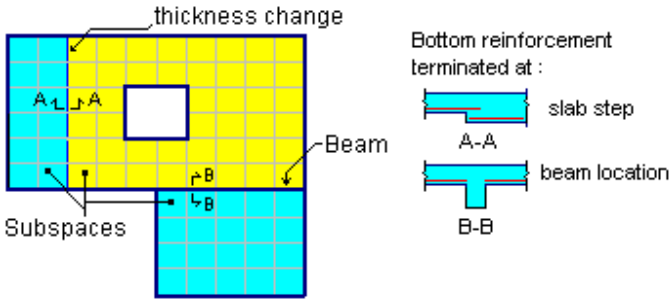
To calculate, detail and draw slab reinforcement:

- select

to define subspaces:
  Subspaces are defined areas on a slab:
  - reinforcement is terminated at subspace boundaries (optional lap into adjacent subspace)
  - each subspace can be assigned with different detailing parameters.

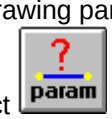
  By default, each level is a separate subspace and this is generally sufficient for most slabs.
  - additional user-defined subspaces may be created along element boundaries, at beam locations and at changes of slab thickness.
  - separate subspaces may be defined for top and bottom reinforcement

  For example:

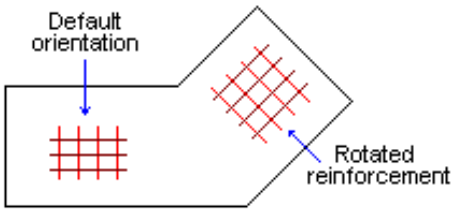


- select

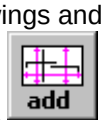
to specify the default parameters for the entire model:
  - reinforcement type: bars or prefabricated meshes
  - steel type, concrete type, cover, etc.
  - bars/mesh parameters: size, min/max diameter, spacing, length, etc.
  - drawing parameters: text size, titles, etc.

- select 

to revise any of the above parameters for selected subspaces. Use this option to place bars at a different angle in a subspace. For example:

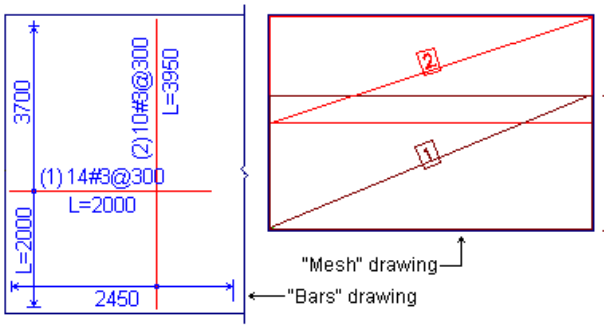


- select

to create drawings and add objects to them
  - select a drawing and click 

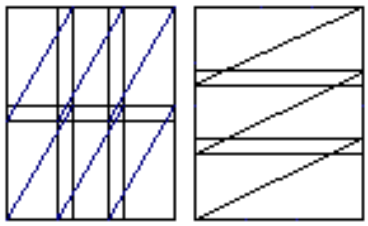
  - add slab reinforcement drawing, bar schedules, mesh schedules or mesh details.
- check the reinforcement arrangement and revise parameters if necessary

Examples:



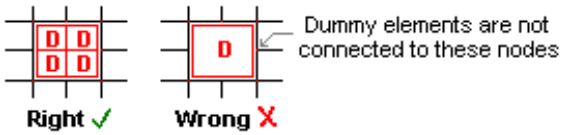
The program details the reinforcement as follows:

- As,req'd is calculated at the center and four corners of each element; the maximum of these values is used for the entire element.
- Bars or meshes are selected according to the parameters.
- The program arranges the reinforcement in two patterns, each with the longer bars in a different direction, and selects the pattern with the smaller weight of steel.



Hints and Suggestions:

- for the "Bars" option: If you decided to place bars only where required but the program details bars over most of the slab area:
  - select **Results** in the toolbar and then **Display detailed results**; note the total weight of reinforcement in the slab.
  - select "**Defaults**" (or "**Parameters**") and set the option to ☑ **Put bars also where As = 0**.
  - display the revised drawing and check the total weight of reinforcement again; the second solution is preferable if the weight increase is relatively small because the bar arrangement is simpler and more uniform.
- if you selected **User defined bars/mesh and additional bars/mesh**:
  - click the "**Edit**" icon and select ◉ **Display only additional bars**
  - if the additional bars/mesh have been detailed over a relatively large area, decrease the spacing and/or increase the diameter of the fixed bars/mesh so that the additional bars are placed only over a relatively small area.
- if there are small openings in the slab, it may be more practical to detail the slab with bars placed over the opening (to be cut away on site) rather than trying to arrange the bars around the opening. Define "dummy elements" over the openings (the area required will always be zero even if minimum reinforcement is requested). Note that the dummy elements must be connected to all nodes on the opening perimeter:



Note:

- in general, the scale should be 1:50. If the drawing size is reduced (e.g. 1:100), the text (bars and dimensions) will overlap and make reading the drawing very difficult. It will probably be necessary to reduce the text size in such cases.

*** Blank page ***

9 Bridge Postprocessor

9.1 Introduction

Most bridge design codes require that each point on the bridge be designed for the arrangement and combination of loads that produce the most adverse moments, shears, etc. at that point.

In order to comply with the requirements of the Code, the bridge designer has to calculate influence lines for each result type at every point along the bridge. Based on the results, he then decides where to load the bridge. It is obvious that as the bridge becomes more complicated, the amount of work required increases enormously.

The *STRAP* bridge module calculates the load patterns and the corresponding results, as follows:

- The user defines a regular *STRAP* model of the bridge consisting of beams and/or elements, all load cases (self-weight, wind, earthquake, etc.) - other than highway (vehicle) loads, and solves the model for these loads.
- The user then selects the **Bridge design** option and defines the following:
  - lanes
  - lane loads
  - load cases
- The program then calculates for every point in the model and for every result type (moment, shear, etc.) the combination of uniform, vehicle and knife-edge loads that give the maximum and minimum result at the point. The calculation is carried out according to user specified parameters, such as the number of lanes that may be loaded simultaneously, the uniform load intensity for various lengths of load , etc.
- The user can then append load cases to the *STRAP* results file containing maximum/minimum results at each point.
- In addition, the user can select any point on the bridge and generate an influence line at that point for any result type. Both the influence lines and the corresponding load arrangement may be displayed graphically.

The basis of the calculation is the division of each lane into a series of strips. The width of each strip (perpendicular to the axis of the lane) is equal to the width of the lane and the length of the strip (parallel to the axis of the lane) is specified by the user. The program loads and solves each strip with a uniform unit load and calculates the influence lines and the critical load arrangements from the results. Similar to finite element analysis, the accuracy will increase as the lanes are divided into smaller strips, but so will the solution time and the disk space required.

9.2 Solution Method and Program Assumptions

The program divides each lane into strips; the width of each strip is equal to the width of the lane at that point while the length of the strip (parallel to the axis of the lane) is defined by the user at the time the lane is defined.

The program calculates the deflections in the entire model for a unit uniform load applied to each strip, e.g. for a model with 500 strips, the program solves 500 unit load cases.

At every beam end (and at every 1/10 span) and for every result type (moment, shear, etc.), the program searches for the strips in every loaded lane where the corresponding result for the unit load on that strip has the same sign as the result being calculated. The number of strips used corresponds to the load length defined. The program then multiplies the results from the unit loads by the load intensity.

A similar method is used for elements; the program calculates the results at the element centre and the corners. The maximum deflection is calculated at nodes, except at nodes that were defined as supports, where the max/min reactions are calculated.

When calculating maximum/minimum results, the program applies the loads to the strips according to the influence line results; loads are applied only on the strips where the applied load gives a result with the proper sign. Note that the maximum load length (i.e. sum of loaded strip widths) may be limited by the user.

9.3 How to use the Bridge Module

STRAP

- Create a model of the bridge in the *STRAP* geometry module.
- Define all loads other than the bridge loads in the *STRAP* loading module.
- Solve the model (optional at this stage, but before *STRAP* results are displayed, design postprocessors are accessed, etc.)
- Return to the *STRAP* Main menu. Select **Design** in the menu bar and **Bridge module** in the pull-down menu.

Bridge Module

- Define lanes:
  Set **lanes** in side menu and click the **Define** icon.
  Define the start and end nodes of each segment, the lane width and the number of strips per segment.
- Select the method of distributing the loads to the model:
  - select **Options** in the menu bar
  - select **Load direction** to specify the Global direction in which all loads are applied.
  - select **Load distribution** in the pull-down menu and specify the element types that the loads are applied to.(beams, specific beams, elements)
- Solve the model:
  Select **Files** in the menu bar and **Solve** in the pull-down menu. The program will return to the bridge module after completing the solution.
- Display influence lines:
  Select **Results** in the menu bar and
  - graphic display: select **Draw influence lines** in the pull-down menu
  - tabular display: select **Display influence lines table** in the pull-down menu
- Define lane loads:
  set **lane loads** in side menu and click the **Define** icon.
  The following load types may be defined:
  - Uniform load:
    Specify the uniform lane load (units are t/m, kip/ft, etc. - not load/area), the maximum length of the sum of the loaded strip widths and the load length reduction factor table.
  - Vehicle load:
    Select a vehicle load or vehicle group from the list box, define a factor to increase/decrease the load and the direction of travel (most vehicle loads are not symmetric). Note that the program can check both directions and use the worst.
  - Knife edge load (also used to define concentrated loads).
- Define load cases:
  Load cases are defined by assigning lane loads to specific lanes.
  Set **load case** in side menu and click the **Define** icon.
  - assign the defined lane load to the defined lanes
  - define permutations; the program will create additional load cases by interchanging the lanes the lane loads are assigned to.

  Note that load cases may be deactivated/activated.
- Display a specific result for any node/beam/element or specific point. The results are the value of the maximum/minimum result and the corresponding loaded strips, location of vehicle loads and location of knife edge loads.
  - select **Results** in the menu bar
  - select **Draw applied loads for selected result** in the pull-down menu
  - specify max or min; select result type, specify location.
- Transfer results to *STRAP*:
  The program appends 'transfer' load cases to the *STRAP* result files.
  - Maximum or minimum results may be selected.

- For the same transfer load case, select either:
  maximum/minimum value for each result type,
  maximum/minimum for a specific result type and corresponding results from same calculated load case.

Note that the transfer for very large models with many lanes, strips and calculated load cases may require several hours (the progress is displayed on the screen).

** Blank page **

10 POSTTEN

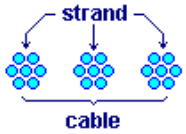
10.1 General

POSTTEN is a *STRAP* postprocessor program that designs post-tensioned beams and slabs in solved *STRAP* models. The design may be carried out according to various design codes.

The *STRAP* model must either be a plane frame or a space frame; grids will not be accepted by the program (axial forces from prestressing cannot be added to load cases).

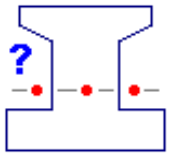
The program uses the following terminology:
- **strand** - the basic prestressing unit, a single wire or a group of wires
- **cable** - a group of strands

For example:



In this example:
- the cable is defined with 3 strands
- the strands are the '7-wire' type.

The following *STRAP* beam sections are accepted by the program:
- all sections defined by "Dimensions", except round sections
- All sections defined by "Properties" (A,I, etc). The program creates an equivalent rectangular section.
- all solid sections defined in *CROSEC* and imported into *STRAP* geometry



Note that the program does not check whether the defined cables actually lie within the section.

Calculation of losses and deflections are calculated at various times stages for each beam measured from the day the beam is cast. All dates (casting and stages) are measured from an arbitrary zero date referred to as the 'start of construction'. For a specific beam:

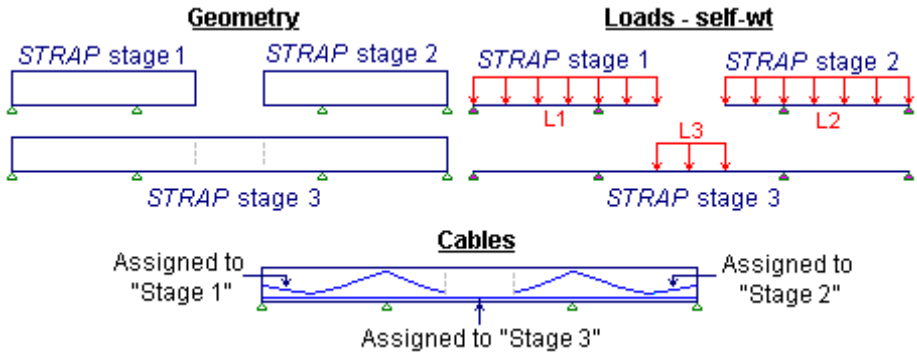
10.2 How to use the program

POSTTEN is a *STRAP* postprocessor program that designs post-tensioned beams and slabs in solved *STRAP* models. The *STRAP* model must be defined as follows:
- Define load cases with all loads, but without prestressing loads
- Solve the model
- Define load combinations in the results module
- Select the **Files - Post-tension** option

To design post-tensioned beams and slabs:
- Specify the design parameters, e.g. concrete type, prestress loss parameters, etc.
- Define a "Stage table" for models where not all of the cables or beams are prestressed at the same time. The stages are named and defined by the number of days from the start of construction.
- Define all of the beams to be prestressed by selecting the start and end *STRAP* member.
- Define all of the slabs to be prestressed by specifying the start and end nodes of prestressing "lines" that pass through the relevant elements. Each "line" is designed similarly to a beam, i.e. all cables are parallel to the line. Note that the lines do not have to be parallel to element boundaries.
- Expand the combination table:
  - specify the "Permanent" loads in the combinations (for the calculation of deflections and time losses).
  - assign each load case to specified stage, if relevant. Note that a load case may either be applied or removed at a stage.

- For each beam:
  - specify the type and number of cables and the prestressing force. The program displays a "Magnel diagram" as a design aid for the selection of the prestressing. (refer to 10.3 - How to define cables)
  - specify the trajectory of the cables. The program displays the upper and lower permissible cable boundary lines superimposed on the beam elevation.
  - assign selected cables to different stages, if relevant.
- select "Solve"; the program creates cable force load cases at each of the defined stages and time steps, including losses at each step, and solves the model for these cases. The new load cases and their results may be viewed in *STRAP.*
- Revise the cable details and trajectory, if necessary.
- Display/print tables of stresses, deflections, shear and ultimate moments for each beam.

If the prestressing is done on several distinct model stages that represent intermediate stages of construction, the stages must be defined in *STRAP* geometry. These *'STRAP* stages' are then assigned to the relevant design stages in the *POSTTEN* module. For example:



STRAP:

- Create the separate stages, then define the loads for each stage (without prestressing loads) in a separate load case and assign it to the relevant stage; solve the model. Note that *each load is defined only once*, at the stage where it is *first applied*. This is illustrated by the self-weight loads in the example above; the self-wt of the cantilevers are applied only in Stage 1 and 2, even though they continue to act in Stage 3.
- Define load combinations in the *STRAP* results module. These combinations should represent the loads acting at each stage. for example: C1=L1, C2=L2, C3=L1+L2+L3

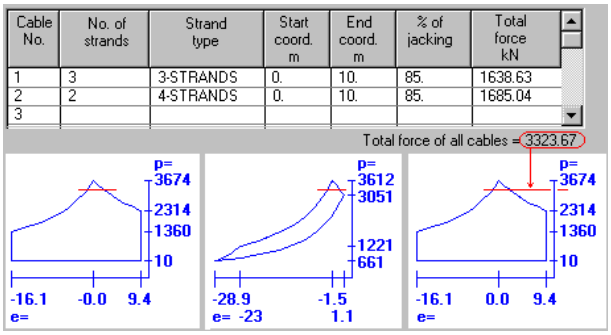
POSTTEN:

- Assign the relevant *'STRAP* stage' to each POSTTEN stage in the **Stages - stages** option.
- Assign the combinations to the relevant stages in the **Stages - Load table** option.
- Define the cable; assign each cable to the relevant stage in the **Design - losses - jacking sequence** option, as shown above.

10.3 How to define cables

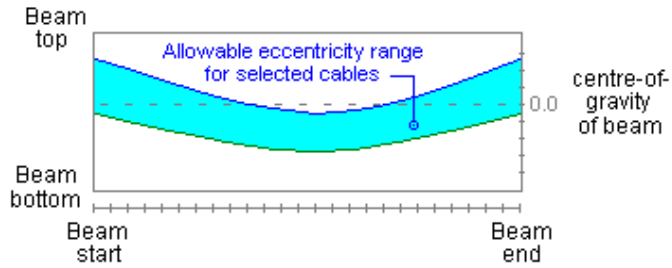
- select

- define cables (each may contain more than one strand) so the total force lies within the Magnel diagrams. For example:



- select

- double-click one of the cables in the table; the program superimposes the minimum/maximum eccentricity range *for this cable*. For example:

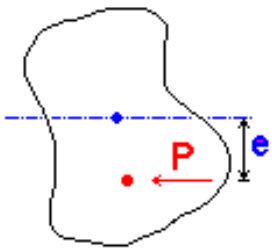


- the cables may be defined with a straight or parabolic trajectory; select one of the options and define the trajectory interactively on the screen; click

when finished.
- select

again and choose another cable. The allowable eccentricity range displayed will again be o*nly for the selected*, i.e. it will reflect the prestressing and eccentricity of all other cables with defined geometry. Placing this cable within the range insures that the total force/eccentricity of all cables provides a satisfactory solution.

10.4 Magnel diagrams



The prestress force **P** and the eccentricity of the force **e** must be selected by the user; any number of combinations of **P** and **e** provide an acceptable solution.

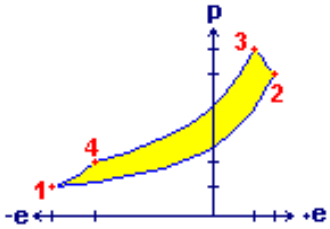
The stresses at the extreme fibres are limited to the Code values and are checked at every stage for maximum and minimum moments with the actual prestressing force (after losses) . This gives four limiting stress conditions:

- minimum moment - top fibre
- minimum moment - bottom fibre
- maximum moment - top fibre
- maximum moment - bottom fibre

The equations are in the form:

$$\sigma_e - \frac{P}{A} \pm \frac{P\,e\,y}{I} \leq \sigma_{all}$$

where σ_e is the relevant stress from the external loads.



Plotting all of the possible solutions for each of the four conditions gives four intersecting lines: 1-2, 2-3, 3-4, 4-1 in the adjacent diagram; any combination of **P** and **e** within the enclosed area will result in acceptable stresses in the specific cross-section.

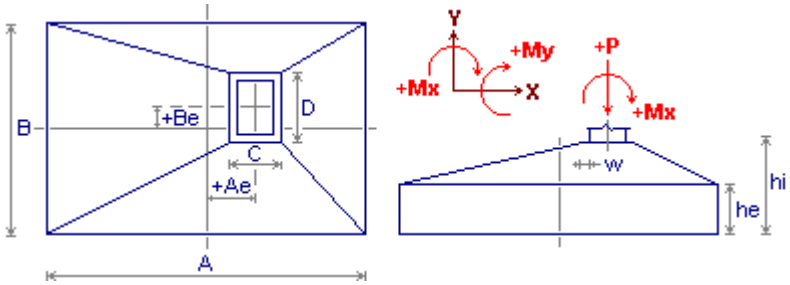
11 Foundation Design

11.1 Program Specifications

Program FOUND designs reinforced concrete spread footings. The program features are:

- design for axial force and biaxial moments (in same load case)
- eccentric column allowed in both directions
- "Quick design" of several footings by entering data in table
- multiple load cases (up to 25)
- automatic design of base dimensions and reinforcement according to Code requirements
- automatic design of height according to punching and shear, including sloped upper surface
- automatic design and detailing of reinforcement according to Code requirements
- user defined dimensions may be checked
- design units may be specified by the user
- graphic output of footing - plan and section - to scale and in colour
- DXF file of graphic output may be created

The following symbols are used by the program



Note the sign conventions for:

- positive eccentricity of the column about the center of the base
- positive applied moments and axial load.

11.2 How to use FOUND

FOUND can be run either as a standalone program or can be run from the *STRAP* result module to design the footings at supports in *STRAP* models.

11.2.1 Stand – alone program

- Specify column dimensions and design parameters:
  Select **Design** in the menu bar and then **Parameters** in the pull-down menu
- Define loads:
  Select **Design** in the menu bar and then **Loads** in the pull-down menu; more than one load case may be defined.

The program will now display a solution based on the specified parameters, loads and dimensions.

- Refine the footing base dimensions:
  Select **Design** in the menu bar and then **Dimensions** in the pull-down menu; enter new footing base dimensions (eccentric footings may be defined at this stage) and the program will check their validity. Each set of dimensions is displayed in the table on a separate line; click on any line to select a solution.
- Refine the footing height dimensions:
  Select **Design** in the menu bar and then **Sloped footing** in the pull-down menu; enter a new footing height (define a different exterior height to create a sloped footing) and the program will check their validity; click on any line to select a solution.
- Print results:
  Select **Output** in the Menu bar and the **Print drawing** or the **Print results** options.

11.2.2 STRAP post – processor

The program designs rectangular spread footings at all nodes assigned with restraints (and springs, if specified by the user).

The postprocessor automatically retrieves from *STRAP*:
- the reactions for all *STRAP* load combinations at these nodes (force and moments)
- the column dimensions (if possible) from the section of the member attached to the node.

Upon selecting the Footing postprocessor option in *STRAP*, the user can choose to design:

All/several footings
The program automatically designs the footings for the group of supports selected. Design parameters may be defined for the entire model. Results for the entire model are displayed in tabular form. Results for footings previously designed with the Single footing option will not be overwritten.

Single footing
The program automatically designs the footing for the support selected. Design parameters may be revised for the individual footing. Load cases may be added, revised or deleted. Results for the individual footing are displayed in graphic form. Results for footings previously designed with the All/Several footing option are overwritten.

Current results for any footing may be viewed by selecting **Single footing** and then selecting the associated node. Refer to 11.2.1.

Note:
- footings are assumed to be oriented according to the local x2,x3 axis of the column
- the program always transfers the column dimensions as a rectangle that bounds the actual section dimensions. The loads are assumed to act in the center of this rectangle.
- for columns defined by properties (A,I), the program transfers zero dimensions and the postprocessor begins the design with a default dimension that may be revised by the user.

12 Section generator - CROSEC

12.1 Program Specifications

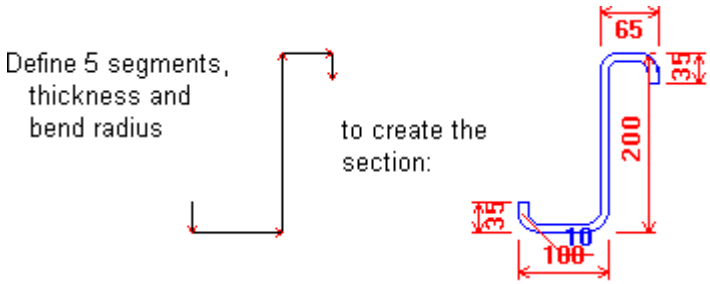
CROSEC is a program that calculates the properties of geometric sections (area, moment-of-inertia, center-of-gravity, etc).

The program can calculate all properties that are required for the design of cold-formed (light gauge) sections. All line sections (with any arbitrary shape) defined in this program may be transferred to the *STRAP* steel postprocessor and can be calculated as cold-formed sections.

Two general types of sections may be defined:

- **Line sections:**

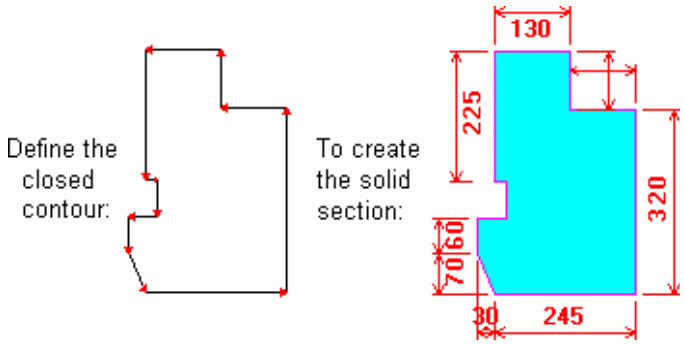
  Sections composed of a series of connected lines, each with a specified thickness. For example:



  - Segments may be defined in any direction (diagonal).
  - Several separate sections may be defined to form a section; the additional sections are called "subsections". Each subsection may have a different thickness and bend radius.
  - The properties of a Line section may be copied to *STRAP* geometry
  - Tables of cold-formed sections may be added to the *STRAP* property tables; the user first defines the dimensionless general shape and then enters the dimensions for each segment of every section in the table. Different segments may be defined as "equal" (only one of the dimensions must be entered).
  - Properties calculated include: Shear center, warping constant, torsional moment-of-inertia and torsional-flexural buckling constant

- **Solid sections:**

  Sections formed by a closed contour. For example:



  - Several separate contours may be defined to form a section; the additional contours may be specified to be "holes".
  - The properties may be copied to the *STRAP* geometry.
  - Standard solid shapes may be defined by entering the dimensions (parametric section library)
  - Solid shapes may be imported from a DXF file.
  - Properties calculated include: Torsional moment-of-inertia.

12.2 How to use the program

- Select **New line section** or **New solid section** in the **Sections** menu in the menu bar

- **New line section**:
  define a series of connected segments to form the section
  New solid section:
  define a closed contour to form the section

- Add subsections, if necessary to create more complex sections. For example:
  New line section: define a series of connected steel sections to form light-gauge steel sections. The subsections are defined separately and must be "connected".
  New solid section: define a hole in the first contour

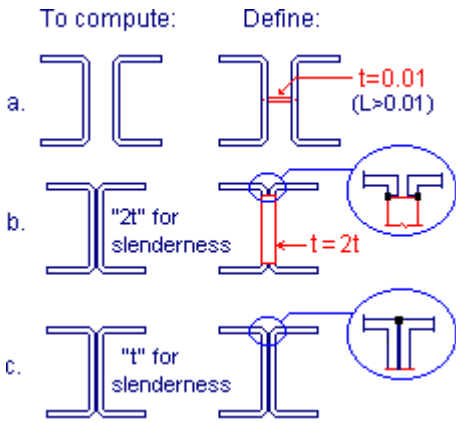
- Select **Display properties** in the **Output** menu in the menu bar to display the computed section properties (A, I, J, centre-of-gravity, etc) and **Print section** to print the results.

- Transfer the properties of the current section to *STRAP* geometry; select **Copy to clipboard** in the **Output** menu in the menu bar

For line sections only:
Create a table of cold formed sections for the *STRAP* properties table. The table is created from a Line section by defining sections with similar shapes but with different dimensions:
- select **Create table** in the **Section table** menu in the menu bar
- select an existing Line section and enter a table of dimensions for the shape.
- select **Transfer to STRAP** in the Section table menu in the menu bar to copy all section in the table to the *STRAP* property files (as cold-formed sections)

12.3 Hints and suggestions

- For separated back-to-back channels, angles, etc; connect the two channels with a subsection having the correct length and a thickness = 0.01 (Example a).
- For connected back-to-back channels, etc, where the slenderness of the web must be calculated using the combined web thickness, define the web as a subsection with the double thickness and the flanges by four subsections connected to the corners of the web (Example b).
- For connected back-to-back channels, etc, where the slenderness of the web must be calculated using the web thickness of a single channel, define the second channel as a subsection and connect at one point (Example c).



A1 Hardware Requirements and Program Capacity

Minimum Hardware: hard disk with 35 mb of free space

Operating System: Windows 98/2000/NT/XP

Capacity:

Program Capacity	Regular Version	Enhanced Version	Professional Version
Maximum node number	1000	2500	32000
Maximum element number	2500	32000	32000
Maximum property group number	200		
Maximum number of loading cases	1000		
Maximum number of combinations	1000		
Maximum number of load cases per combination	1000		
Maximum number of load groups	100		
Maximum number of load cases per group	1000		
Maximum DOF bandwidth after optimization	16000		
Maximum number of mode shapes	1000		

Note:

- *STRAP* may crash if other programs with large memory demands (e.g. AutoCAD) are running simultaneously.

A2 Printing the Manual

The entire *STRAP* manual is supplied with the program in 'PDF' (Adobe Acrobat) format files. All text in the program "Help" is included in the printed manual.

The files are located on the installation CD in folder **\STRAP\DOC**.

To print the manual, use the Adobe Acrobat Reader program (copy from the *STRAP* Installation CD or download from the Adobe Internet site).

The manual is contained in a series of files:

File:	**Contents**:
STRAP.PDF	STRAP manual, except for the following:
CODE_UK.PDF	BS5950, BS8110
CODE_US.PDF	AISC, AASHTO, AISI, ACI318
CODE_CDN.PDF	CSA S16.1, A23.3, C136
CODE_EUR.PDF	Eurocode2, Eurocode 3, Eurocode 4
CODE_IND.PDF	IS:800, IS:456
CODE_CHN.PDF	GBJ17
FOUND.PDF	Foundation design program
CROSEC.PDF	Section generator program
CMD_MODE.PDF	Command mode (batch) manual
VERIF.PDF	Verification manual
DEMO_USA.PDF	Demo and Tutorial examples - American units (ft-kip)
DEMO_MTR.PDF	Demo and Tutorial examples - Metric units (kN-meter)
SHORT_MAN.PDF	This manual (Condensed *STRAP* manual)
SMAN_EX.PDF	This manual - Examples